# Structural Analysis Software Applications

Science and Technology Support
High Performance Computing

Ohio Supercomputer Center
1224 Kinnear Road
Columbus, OH 43212-1163

# Table of Contents

Introduction
LS-DYNA Program
ABAQUS Program
ANSYS Program

# Introduction

Course Objectives

OSC Structures Packages

OSC Computer Systems

Access to OSC Systems

Enabling X-Window Graphics

# Course Objectives

1) Provide the user with information necessary to select which software package is best suited for a given problem.

2) Review basic functionality of each package

This will be accomplished by:

- ⇒ Review the modeling capabilities of each package
- ⇒ Learn how to configure graphic sessions from OSC to your local workstations
- ⇒ Examine the basic functionality of pre and post processors through lecture, examples and labs
- ⇒ Learn how to execute analysis in batch mode
- ⇒ Identify important sources of documentation and information

# OSC Structures Packages

ABAQUS

ABAQUS/Standard
ABAQUS/Explicit
ABAQUS/Post
ABAQUS/Viewer
ABAQUS/CAE

The MacNeal-Schwendler Corporation

NASTRAN

ANSYS

ANSYS 5.5

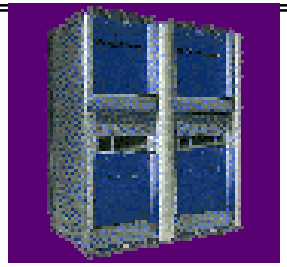
Livermore Software Technology Corporation

LS-DYNA 3D
LS-NIKE 3D
LS-INGRID
LS-TAURUS
LS-POST

# Overview of OSC Systems

CrayT90

High end vector processing
Shared memory - 1 Gbyte
4 processors



Origin 2000

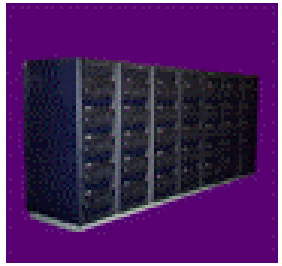
Scalable RISC system
Distributed/Shared memory
16 Gbytes
32 processors

Cray SV1

Scalable vector processing
Shared memory - 16 Gbytes
16 processors

CrayT3E

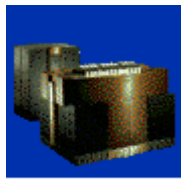
Massively parallel system
High performance components
Distributed memory - 16 Gbytes
128 Processors



Cluster

Massively parallel system
Commodity components
Distributed memory - 64 Gbytes
128 Processors

# Software by System




Cray T90 (osca.osc.edu) - Shared memory vector machine

ABAQUS v5.7
LS-DYNA v 940.1a & LS-DYNA v 950
NASTRAN v 70.5
ANSYS v 5.5

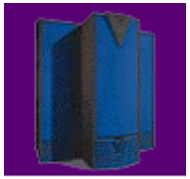
Cray T3E (t3e.osc.edu) - Distributed memory MPP machine

MPPDYNA v 940.0 beta

SGI Origin 2000 (origin.osc.edu) - CCNUMA memory machine

ABAQUS v5.7-4, v5.8-1and v5.8-17
LS-DYNA v 940.0, 940.1a, 950.0
ANSYS v 5.5

# Software by System


Cray SV1

Cray SV1 (oscb.osc.edu) - Shared memory vector machine

LS-DYNA v 950

Cluster

OSC/SGI Intel Linux Cluster (oscbw.osc.edu) - Distributed memory MPP machine

LS-DYNA v 940.2

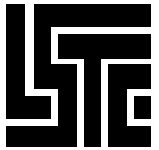
# Access to OSC systems

- All access is remote
  telnet, ssh
- Local machine => (the machine you are sitting at)
- Remote machine => The OSC supercomputer (T90, T3E or Origin)
- X-Window Graphics
  - Standard on Unix workstations
  - Need extra software for PC or Mac
- Pre and post processors may be run interactively
- Solvers must be run in batch mode

# Enabling X-Window Graphics

| Command | Description |
|---|---|
| `xhost + origin.osc.edu` | -This command is executed on your *local* machine. It authorizes the OSC computer to display graphics on your *local* workstation. You must issue this command for each computer you want to use. Only for UNIX workstations. |
| `w -W (on Origin 2000)`<br>`who (on T90 and T3E)` | -This command is executed on the *remote* machine. The purpose is to get your *local* IP address.<br>`jimg ttyq32 3:48pm`<br>`oscnet108.osc.edu` |
| `echo $SHELL` | `-"echo $SHELL" will tell you which shell you are in` |
| *`if csh`*<br>`setenv DISPLAY oscnet108.osc.edu:0.0`<br>*`if ksh`*<br>`export DISPLAY=oscnet108.osc.edu:0.0` | -This command is executed on the *remote* machine. It instructs the *remote* computer where to send the display. The *local* name was obtained from the 2nd line printed by the 'w -W' command. |

# LS-DYNA Package



LS-DYNA Modeling Features

LS-DYNA Components

LS-DYNA Documentation

Model Generation with LS-INGRID

Running LS-DYNA Solver on T90 & SV1

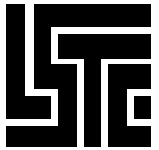
Running LS-DYNA Solver on T3E

Running LS-DYNA Solver on Origin

LS-TAURUS Post Processor

LS-POST Post Processor

# LS-DYNA Modeling Features

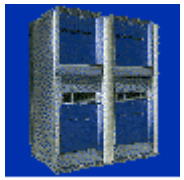


Nonlinear dynamics
Rigid multi-body dynamics
Quasi-static simulations
Thermal analysis
Fluid analysis
Eulerian capabilities
ALE (Arbitrary Lagrangian-Eulerian)
Fluid-structure interactions
FEM-rigid multi-body dynamics coupling (MADYMO, CAL3D)

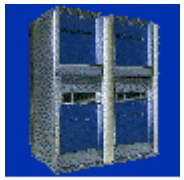
Crack propagation
Real-time acoustics
Design optimization
Implicit springback
Multi-physics
Structural thermal coupling
Adaptive remeshing
Failure analysis
Underwater shock

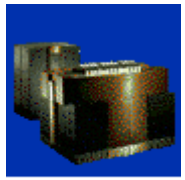
# LS-DYNA Components

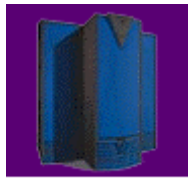
## Pre Processor: LS-INGRID



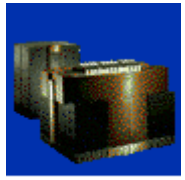

## Solver: LS-DYNA

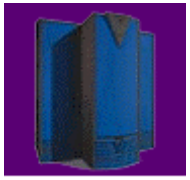




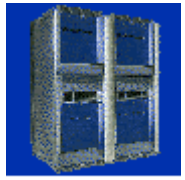


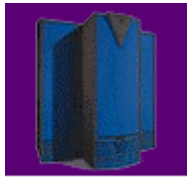
## Post Processor: LS-TAURUS



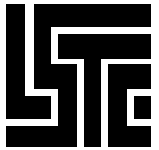



## Post Processor: LS-POST





# LS-DYNA Documentation



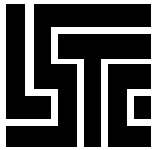
- Documentation is available on-line in Adobe .pdf format

  `http://oscinfo.osc.edu/software/`
  Click on Origin 2000
  Click on LSDYNA

  Manuals are the same for all platforms

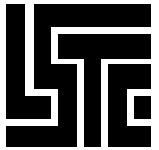
| |
|---|
| LS-DYNA User's Manual (Structural version) |
| LS-DYNA User's Manual (Keyword version) |
| LS-DYNA Theory Manual |
| LS-INGRID User's Manual |
| LS-INGRID Examples Manual |
| LS-TAURUS User's Manual |
| LS-POST User's Manual |

# General Analysis Procedure



- Generate LS-INGRID input file with text editor
  - Define mesh, loads and constraints
  - Less complex than writing straight LS-DYNA input deck
  - Utilities for creating simple cylinders & blocks
- Run LS-INGRID and generate LS-DYNA input deck
  - View mesh
  - Merge coincident nodes
  - Check boundary conditions and applied loads
  - Write out LS-DYNA input deck
- Run LS-DYNA solver
- View results with LS-POST or LS-TAURUS post processor
  - Reads in binary data files generated by LS-DYNA
  - Wide range of display options as well as animation capabilities
  - All calculated properties written to data files.....you select what to display
  - There are some ascii data files you can look through

# Model Generation with LS-INGRID



7 fundamental concepts

- Object space
- Index space
- Reduced index space
- Index progression
- Moving points, edges and planes
- Boundary conditions
- Defining and applying loads

The approach to building a complete model is to break it up into rectangular or cubic blocks of nodes called regions. These regions can then be combined to form the complete physical model.

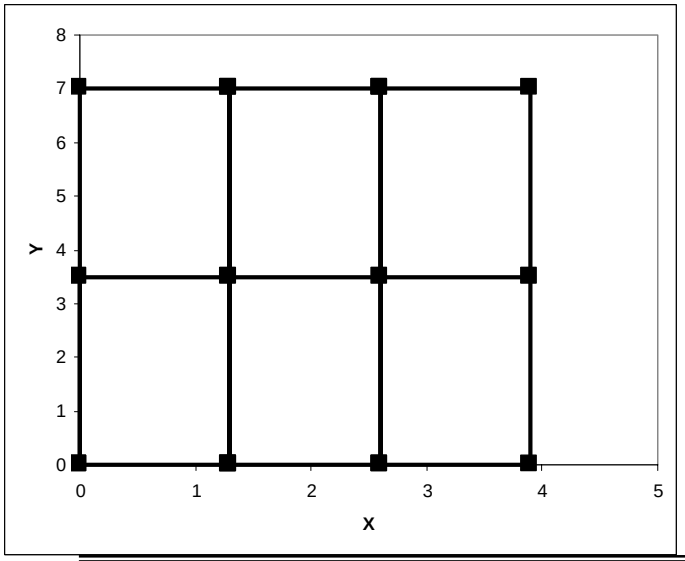
# Object and Index Space

Object space

- Dimensions in physical space
- Specified as: xmin, ymin, zmin, xmax, ymax, zmax

Index space

- Mesh attributes
- Specified as: imin, jmin, kmin, imax, jmax, kmax

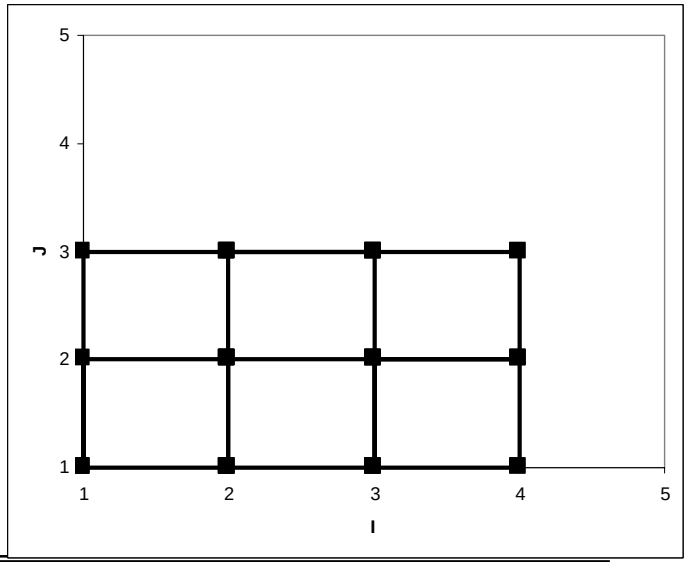



Object space:

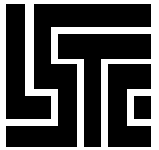
0.0 0.0 3.9 7.0

Index space:

1 1 4 3



# Index Progression

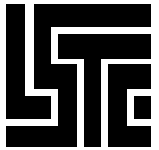


- Builds on the concept of an index region
  imin jmin kmin imax jmax kmax
- This defines the “outside” dimensions of the region, but what if we want to detail an internal component?
  i.e. refine the mesh
- To change meshing in x-coordinate direction:

| | |
|---|---|
| Index region | imin imax |
| | xmin xmax |
| Index progression | imin i2 i3 i4 imax;jmin jmax; kmin kmax; |
| | xmin x2 x3 x4 xmax ymin ymax zmin zmax |

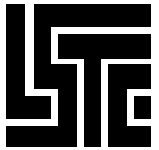
- Also gives the functionality to create multiple regions
- To set min=max, use a negative value
- A zero index is used to indicate that a structure is discontinuous

# Reduced Index Space



- Forces and boundary conditions for the complete object are applied when the region where the force is applied is defined
- Reduced index space means you are narrowing your focus to only the index range of one particular part
- For a generic part imin i1 i2 i3 imax;
  We have reduced index # 1 2 3 4 5

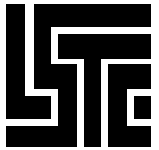
# Moving Regions



- `mb  <region>  n  dx  dy  dz`
- `<region>` is defined using the reduced index
- n is a flag indicating which coordinate to change

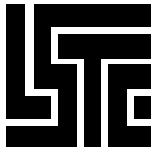
  x - x coordinate  xy - x and y coord.
  y - y coordinate  xz - x and z coord.
  z - z coordinate  yz - y and z coord.
  xyz - x y and z coordinates are to be changed
- Only the coordinates required by flag n need be input.
- dx dy and dz are “added” to the existing x, y, z

# Rotating Regions



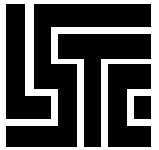
- `rr <region> n theta;`
- `<region>` is defined using the reduced index
  - n = rx rotate about the x axis
  - n = ry rotate about the y axis
  - n = rz rotate about the z axis
- theta is the angle in degrees that you wish to rotate

# Boundary Conditions



- Objects we create on the computer are not infinite in size, they have “boundaries” that define their size
- Setting the conditions at these boundaries is called applying boundary conditions, or B.C.
- There are many types of B.C.
  - displacement, pressure, temperature, velocity, etc.
- We are concerned with two types of B.C. or constraints for our problem
  - displacement (movement in the x,y or z direction)
  - rotation (part will rotate about a given point)

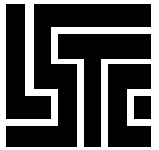
# Boundary Conditions



- The command is ‘b’ and it goes between the ‘start’ and ‘end’ of a part description, but after the index and object space lines
- `b <region> n`
  - `<region>` is defined using the reduced index
- n is a 6 digit binary number

  1st digit: x-displacement; =0 - free, =1 - fixed
  2nd digit: y-displacement
  3rd digit: z-displacement
  4th digit: x-rotation
  5th digit: y-rotation
  6th digit: z-rotation

# Applied Loads



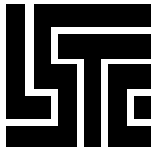
- There are 2 steps to applying a load
  - Define the load curve
  - Apply the load to a given part
- Load curve gives the magnitude of the force over time
- The load curve is ‘defined’ before the first part is defined
- Command

```
lcd  load#  npts  t1  force1  t2  force2 ...
```

where

`load#` is the load curve number
`npts` is the number of time/force data points
`t1...tn` are the time points
`force1...forcen` are the force values

# Applying the Load Curve to a Part



- The load curve is applied between the ‘start’ and ‘end’ of a part definition
- Command

  `fc <region> load# mult <direction>`

  where

  `<region>` is defined using the reduced index

  `load#` is the load curve number

  `mult` is a multiplier (say if you want to double the force)

  `<direction>` indicates which direction the force should act

  format is xdir ydir zdir

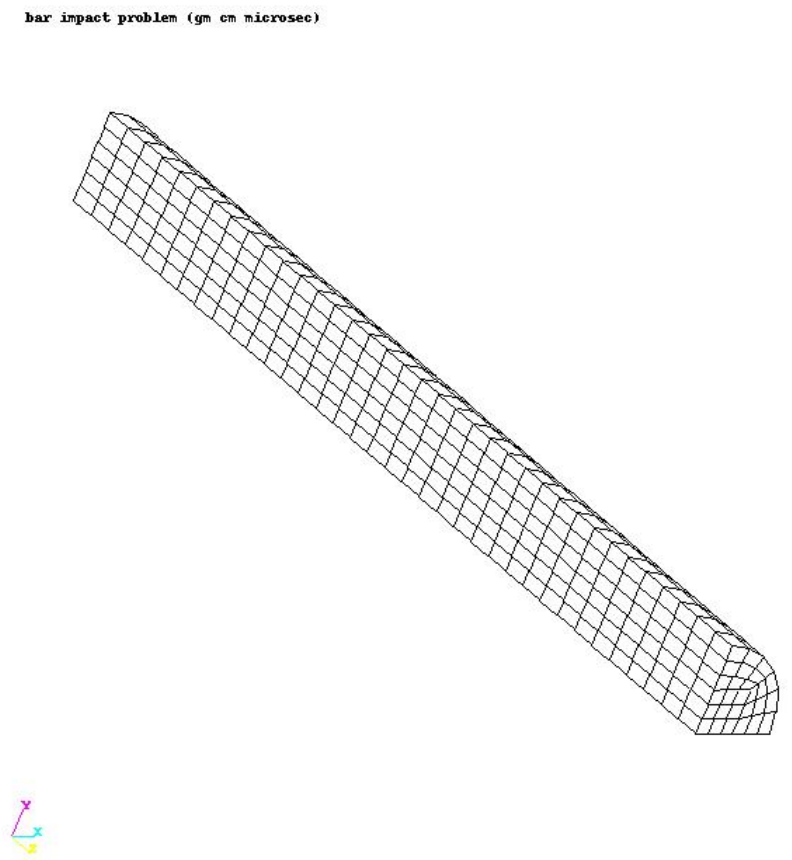
  `1 0 0` would act in the positive x direction

  `0 -1 0` would act in the negative y direction
- load curve can apply over an area or as a point load

# Sample LS-INGRID Input File

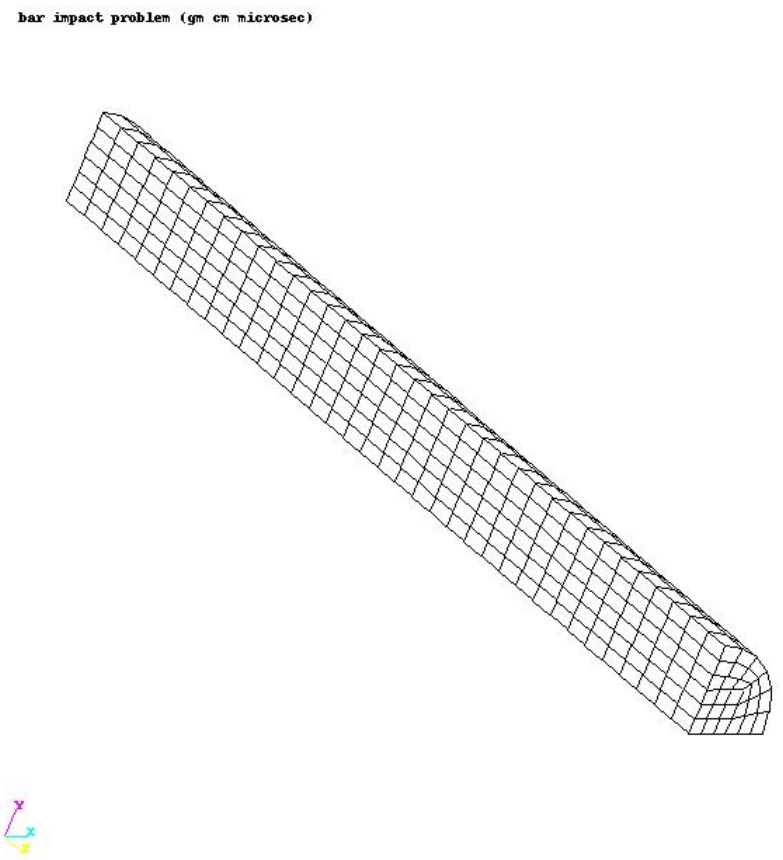
Problem: Circular bar impacting solid surface

```
bar impact problem (gm cm microsec)
 dn3d
 term 80.0
 plti 1.0
 prti 81.
 velocity 0 0 -0.0227
 c Define symmetry planes
 plane 3 0 0 0 0 -1 0 .001 symm
         0 0 0 -1 0 0 .001 symm
         0 0 0 0 0 1 .001 symm
 c Define bar
 start
 c Define index space
 1 4 7 10 13;1 4 7 10 13;1 37;
 c Define coordinate of index space
 -.16 -.16 0 .16 .16 -.16 -.16 0 .16 .16 0 3.24
```



# Sample LS-INGRID Input File

```
c Delete corners for cylindrical mapping
 d 0 1 0 0 3 0
 d 1 0 0 3 0 0
 di 1 2 0 4 5;1 2 0 4 5;;
c Capture surfaces to be mapped
c and map in cyld. space
 sfi -1 -5;-1 -5;;
 cy 0 0 0 0 0 1 .32
 mate 1
 end
 c Use tp 0.001 interactively in Ingrid
 c Define material properties
 mat 1 3
 ro 8.93
 sigy 0.004
 e 1.17
 etan 0.001
 beta 1.0
 pr 0.33
 endmat
 end end
```



# Running LS-INGRID

Only available on Origin 2000

Command: `lsingrid`

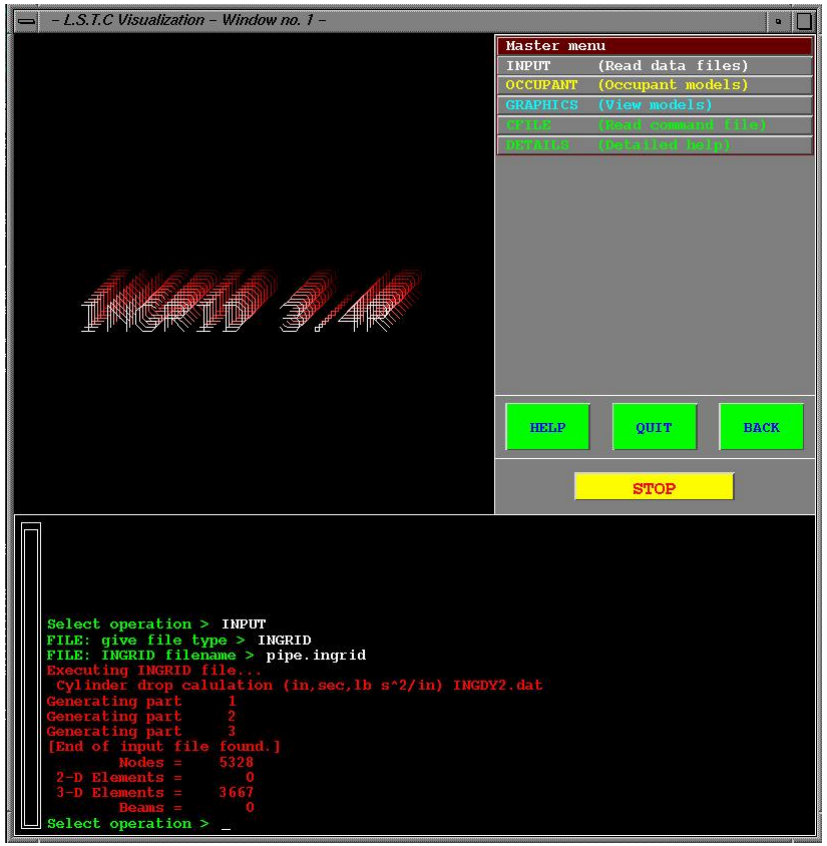
- When asked to choose display device, select ‘X’ for X_Windows
- To load LS-INGRID input file:

Click on INPUT

Select INGRID File Type

Type in the name of your input file

Click on GRAPHICS



# LS-INGRID View Options

IUPD - redraws the object anytime you make a change

GOOD - draws the model in line mode using a hidden surface removal algorithm

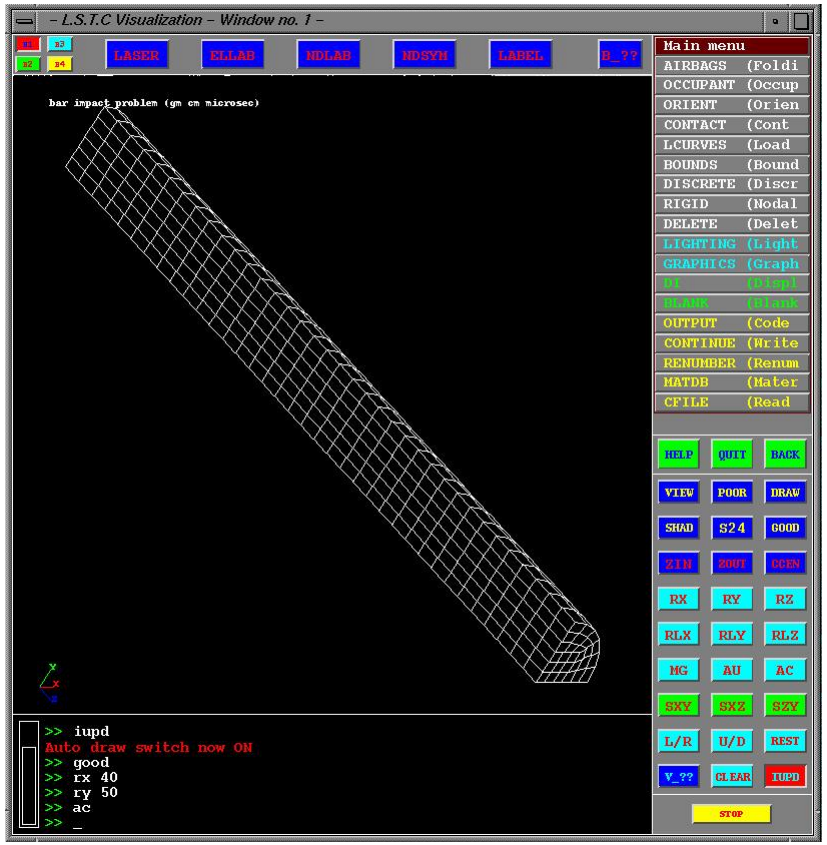
SHAD - gives a solid shaded image

RX, RY, RZ - rotate object about the x, y, or z axis respectively

ZIN - user selected zoom in

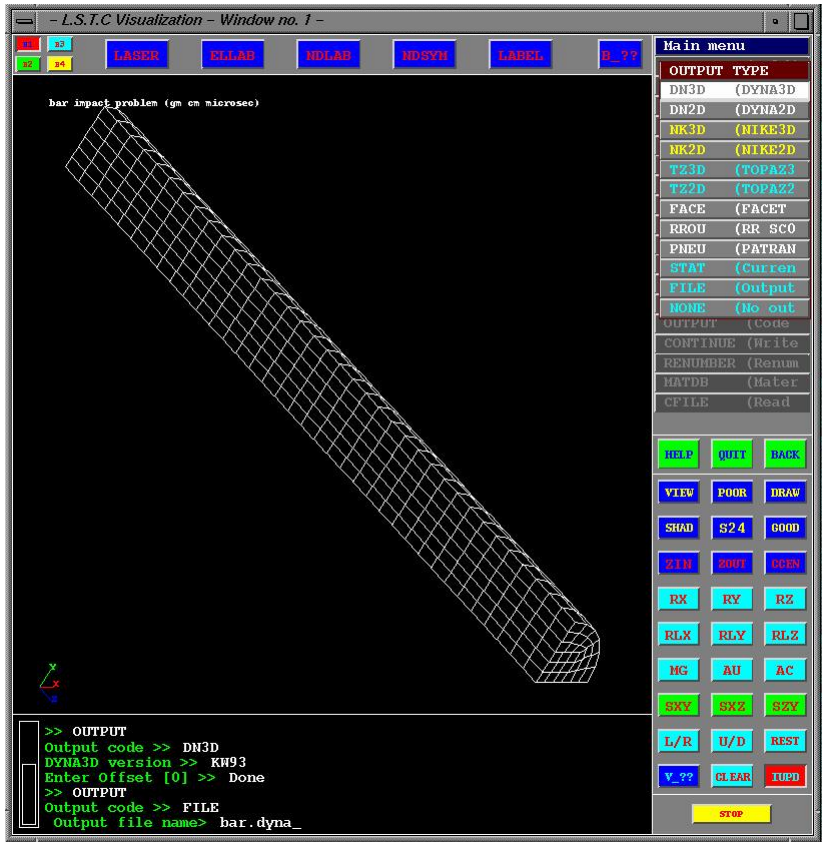
AC - scale the object to fit the entire screen

REST - restore the initial viewing settings

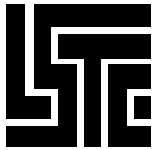


# Generating LS-DYNA Input Deck

- Merge coincident nodes if necessary
  `tp tolerance`
  for this example
  `tp .001`
- Under Main Menu click OUTPUT
- Under Output Type click DN3D
- Under Dyna3d click KW93
- Under Set Offset click DONE
- Under Main Menu click OUTPUT
- Under Output Type click FILE
- Type in the name for the new LS-DYNA input deck
- Under main Menu click CONTINUE



# Running LS-DYNA Solver on T90 & SV1

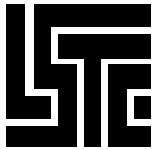


- LS-DYNA supported on all platforms
- T90 & SV1 require the `module load` command
- `ja` command provides a detailed resource usage report including maximum memory used
- Must specify a minimum of 16Mw
- Versions available
  - -v 940.1 (T90)
  - -v 940 (SV1)
  - -v 950

Batch submission file

```
#QSUB -r lsdyna_bar
#QSUB -lT 300
#QSUB -lM 16Mw
#QSUB -eo
set -x
module load lstc
cp $HOME/examples/lsdyna_bar/bar.dyna $TMPDIR
cd $TMPDIR
ja
lsdyna -v 950 I=bar.dyna O=bar.out
ja -clths
cp * $HOME/examples/lsdyna_bar
```

# LS-DYNA Command Line Options



```
lsdyna I=inf o=otf G=ptf D=dpf F=thf T=tpf A=rrd M=sif J=jif
   S=iff Z=isfl L=isf2 B=rlf W=root E=efl X=scl C=cpu K=kill
   V=vda Y=c3d {KEYWORD} {THERMAL} {COUPLE} MEMORY=wds
```

The commonly used options are:

inf = input file (user specified)

otf = high speed printer file (default=D3HSP)

ptf = binary plot file for graphics (default=D3PLOT)

dpf = dump file for restarting (default=D3DUMP)

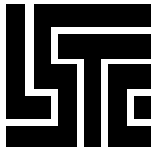
rrd = running restart dump file (default=RUNRSF)

sif = stress initialization file

nwds = number of words to be allocated

If you use the MEMORY command line option, you must request an additional 5Mw in your batch script above what is given on the command line

# Running LS-DYNA Solver on T3E



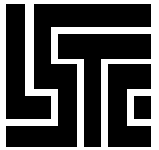
- T3E requires the `module load` command
- `ja` command provides a detailed resource usage report including maximum memory used
- T3E jobs get full use of each node assigned - therefore do not specify any memory limit
- LS-DYNA uses MPI on the T3E
- Note difference in command line

```
mppdyna -np <np> <executable> <options>
```

Batch submission file

```
#QSUB -r lsdyna_pipe_8pe
#QSUB -l mpp_p=8
#QSUB -l mpp_t=700
#QSUB -eo
set -x
module add lstc
cp $HOME/examples/cylinder_8PE/pipe.dyna $TMPDIR
cd $TMPDIR
ja
mppdyna -np 8 I=pipe.dyna O=pipe.out
ja -clths
cp * $HOME/examples/lsdyna_cylinder_8PE
```

# Running LS-DYNA Solver on T3E



Certain LS-DYNA runs may produce ASCII output files. They are first produced as a series of files:

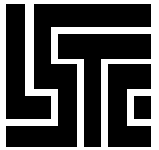
```
dbout.0001
dbout.0002
  .
  .
dbout.nnnn,
```

one for each CPU used in the analysis. To assemble the correct output, first cat the dbout files and then process the result with `dumpdbd`

```
cat dbdout* > dbtotal
dumpdbd dbtotal
```

This will create the ASCII files: glstat, elout, nodout, etc. as specified in the LS-DYNA input.

# LS-DYNA Input Deck Parallel Modification



To run a parallel LS-DYNA job, you must modify the input deck

LS-DYNA3D Keyword file:

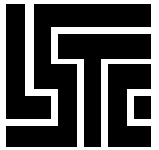
```
*CONTROL_PARALLEL
    NCPU        NUMRHS ACCU
```

| Variable | Description |
|---|---|
| NCPU | Number of cpus used |
| NUMRHS | Number of right-hand-sides written |
| | 0=same as NCPU (recommended for better parallel perf.) |
| | 1=write one only |
| ACCU | Accuracy flag for parallel solution |
| | 1=on (default) |
| | 2=off, faster solution |

LS-DYNA3D Formatted file:

```
$ CARD 16: Computation Option -- Parallel and Subcycling
$...:....1....:....2....:....3....:....4....:....5....:....6....:....7....:....
$NCPU/SORT/SUBS/
     24
```

# Running LS-DYNA Solver on Origin 2000



- Origin does *not* requires the `module load` command
- `ssusage` command provides resource usage including maximum memory used
- Must specify a minimum of 16Mw
- Modifications for parallel job
  - Add "-l mpp_p=#" specifier
  - Add *CONTROL_PARALLEL keyword line to dyna input deck

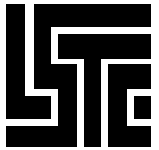
Batch submission file (serial)

```
#QSUB -r lsdyna_bar
#QSUB -lT 300
#QSUB -lM 16Mw
#QSUB -eo
set -x
cp $HOME/examples/lsdyna_bar/bar.dyna $TMPDIR
cd $TMPDIR
ssusage lsdyna I=bar.dyna O=bar.out
cp * $HOME/examples/lsdyna_bar
```

Batch submission file (parallel)

```
#QSUB -r lsdyna_bar
#QSUB -lT 300
#QSUB -lM 16Mw
#QSUB -l mpp_p=8
#QSUB -eo
set -x
cp $HOME/examples/lsdyna_bar/bar.dyna $TMPDIR
cd $TMPDIR
ssusage lsdyna I=bar.dyna O=bar.out
cp * $HOME/examples/lsdyna_bar
```

# LS-TAURUS Post Processor



- Available on T90, SV1 and Origin
  - Recommend running on Origin - better interactive response
  - Avoid running on the T90 if possible - very limited memory
- There are two important command line options

```
ls-taurus_o g=ptf $
```

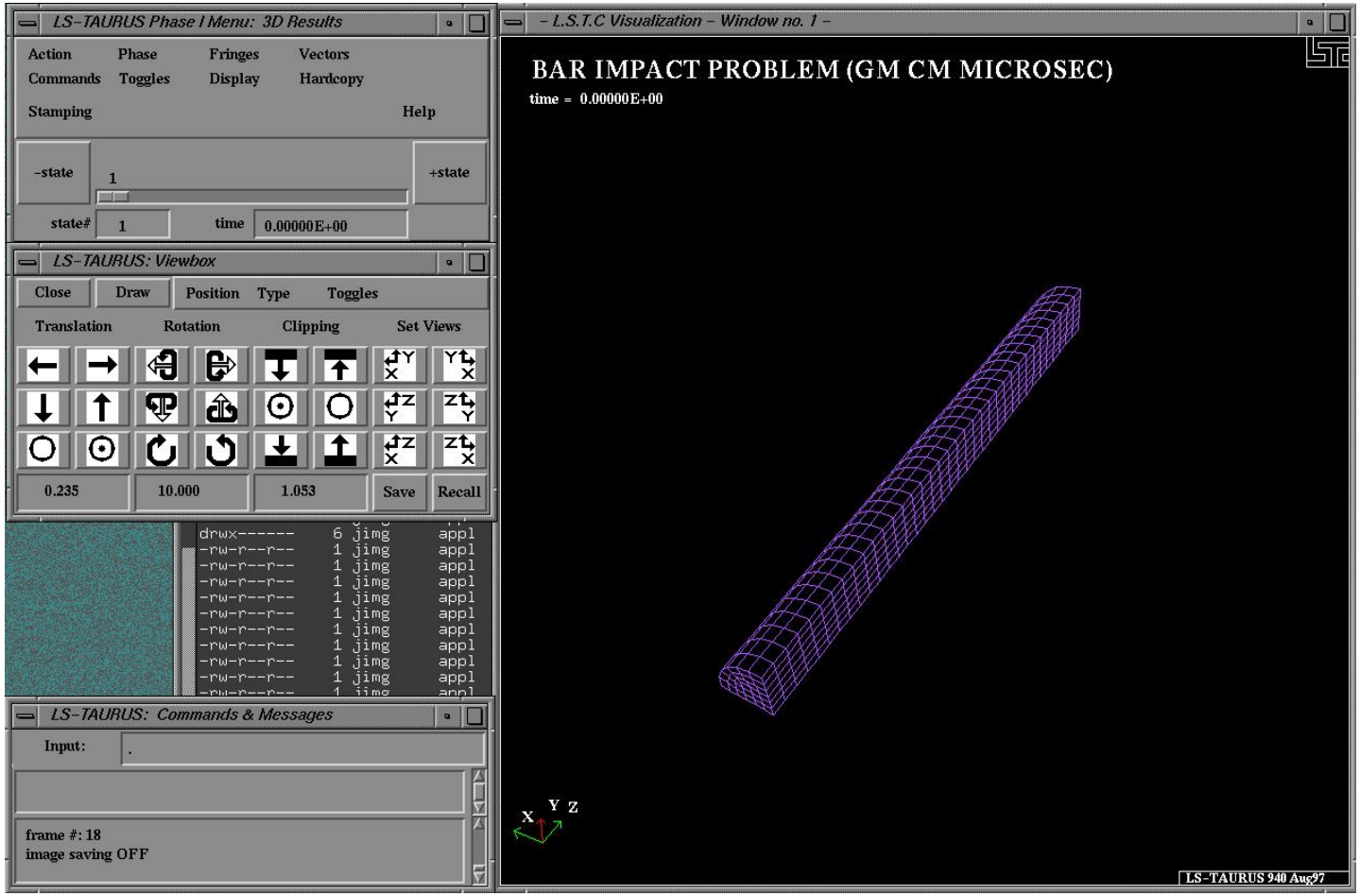
  ptf = binary plot file (d3plot is the default)
  $ allows 32 bit machine (i.e. Origin) read binary files written by 64 bit machine (i.e. T90 and T3E)

- To run LS-TAURUS and read in the default data files

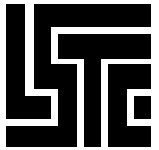
```
ls-taurus_o g=d3plot
```

- At startup you get 4 windows
  - Visualization Window
  - Phase 1 Menu
  - Viewbox
  - Commands & Messages

# LS-TAURUS Post Processor



# LS-Post Post Processor

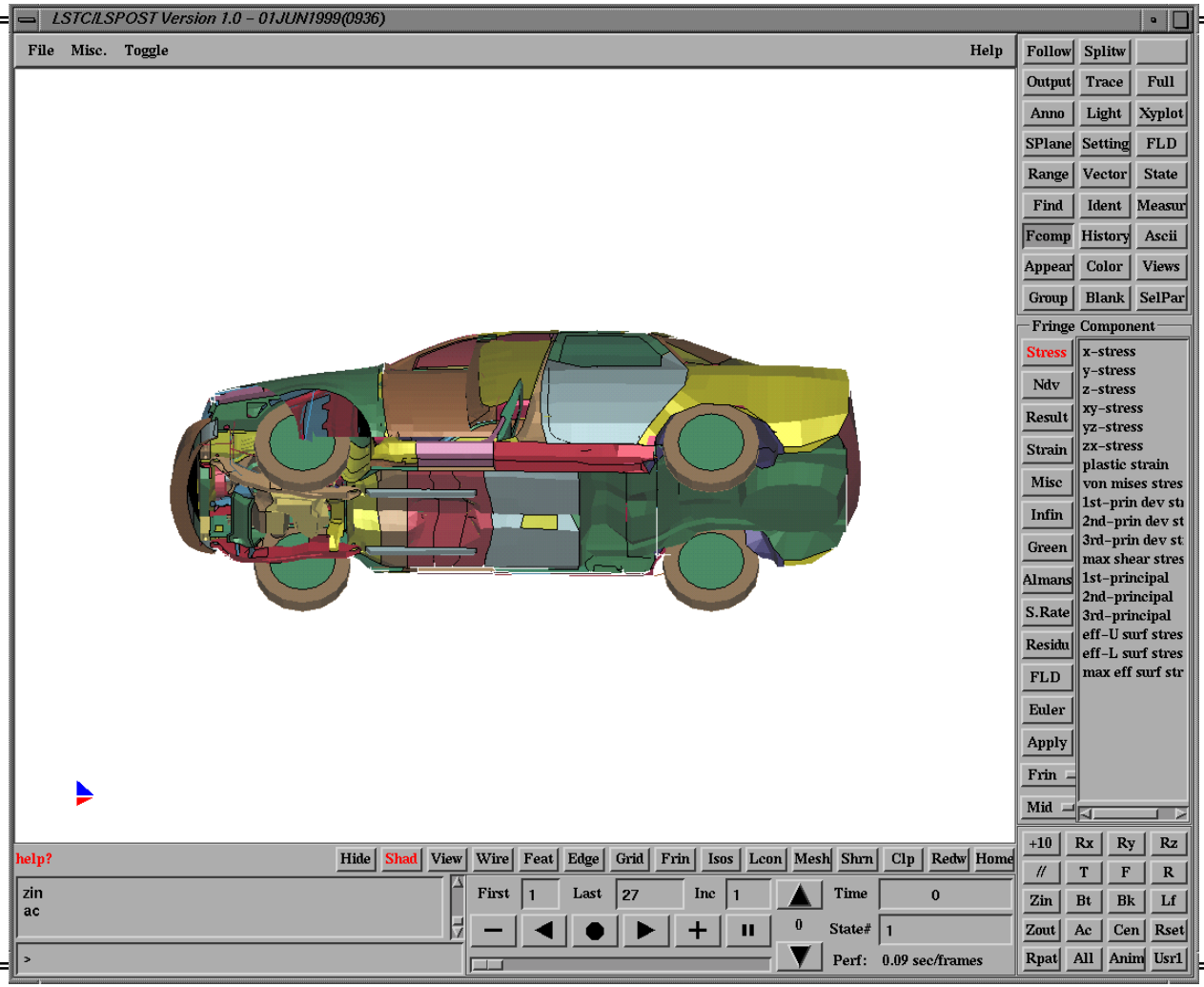


- New post processor available from LS-DYNA
- Available on SV1 and Origin
  - Recommend running on Origin - better interactive response
- There are two important command line options

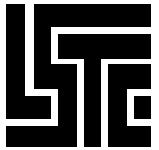
```
ls-post
```

- At startup you get 1 window
  - FILE -> OPEN
    select these menus to read in the d3plot file

# LS-Post Post Processor



# LS-DYNA Web Information



LSTC web site

http://www.lstc.com

Official OSC supported software page

http://oscinfo.osc.edu/software

University of Cincinnati Center of Excellence in DYNA3D Analysis

http://www.ase.uc.edu/~atabiei/center.html

# ABAQUS Package

ABAQUS Modeling Features

ABAQUS Components

ABAQUS Documentation

Generating ABAQUS Input File

Running ABAQUS Solver on Origin

Running ABAQUS Solver on T90

Viewing Results with ABAQUS/Post

Viewing Results with ABAQUS/Viewer

# ABAQUS Modeling Features

ABAQUS/Standard is a general-purpose, production-oriented finite element program. It provides a variety of time- and frequency-domain analysis procedures. These procedures are divided into two classes: "general analyses," in which the response may be linear or nonlinear, and "linear perturbation analyses," in which linear response is computed about a general, possibly nonlinear, base state.

General Analyses

- Static stress/displacement analysis
- Viscoelastic/viscoplastic response
- Transient dynamic stress/displacement analysis
- Transient or steady-state heat transfer analysis
- Transient or steady-state mass diffusion analysis
- Thermo-mechanical (sequentially or fully coupled)
- Thermo-electrical
- Pore fluid flow-mechanical
- Stress-mass diffusion (sequentially coupled)
- Piezoelectric (linear only)
- Acoustic-mechanical (linear only)

Linear Perturbation Analyses

Static stress/displacement analysis:
Linear static stress/displacement analysis
Eigenvalue buckling load prediction
Dynamic stress/displacement analysis:
Determination of natural modes and frequencies
Transient response via modal superposition
Steady-state response resulting from harmonic loading
Response spectrum analysis
Dynamic response resulting from random loading

# ABAQUS Modeling Features

ABAQUS/Explicit is designed specifically to serve advanced, nonlinear continuum and structural analysis needs. The program addresses highly nonlinear transient dynamic phenomena and certain nonlinear quasi-static simulations. ABAQUS/Explicit uses explicit time integration for time stepping and includes the following types of analyses:

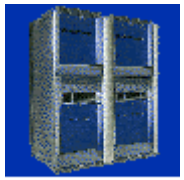
- Nonlinear dynamic stress/displacement analysis
- Explicit dynamic response with or without adiabatic heating effects.
- Annealing for multi-step forming simulations.

Material Definitions

- Elasticity:
  Linear, hyperelasticity, viscoelasticity
- Plasticity and creep:
  Metal plasticity, ductile failure, crushable foam, brittle cracking, brittle failure, strain-rate-dependent plasticity and high-strain-rate failure

# ABAQUS Components

Version 5.8-1
5.8-17




Pre Processor: ABAQUS/Pre

ABAQUS/CAE (Origin only)

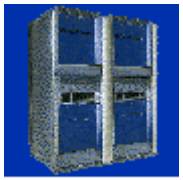
Post Processor: ABAQUS/Post

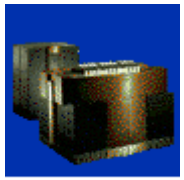
ABAQUS/Viewer (Origin only)

Solver: ABAQUS/Standard

ABAQUS/Explicit

Version 5.7-4



# ABAQUS Documentation

- Hardcopy Manuals are available for reading at OSC and can be purchased from HKS, Inc.
- Online documentation is available on the Origin (v5.8 only)

| | |
|---|---|
| ABAQUS Theory | |
| ABAQUS Standard | |
| Getting Started | (No online version) |
| Users Manual | (Volumes I, II, III) |
| Sample Problems | (Volumes I, II) |
| ABAQUS Explicit | |
| Getting Started | (No online version) |
| Users Manual | (Volumes I, II) |
| Sample Problems | |
| ABAQUS Viewer | |
| ABAQUS CAE | |

# General Analysis Procedure

- Generate ABAQUS input file with text editor
  - Define mesh, loads and constraints
  - Utilities for creating simple cylinders & blocks
- Run datacheck on input files
- View model with ABAQUS/Viewer
  - View mesh
  - Check boundary conditions and applied loads
- Run ABAQUS solver
- View results with ABAQUS post processor
  - Wide range of display options as well as animation capabilities

NOTE: as of the spring 2000, ABAQUS/CAE is under evaluation and only one license has been purchased

# ABAQUS/Pre

Input file is split into two parts:

- *Model data* defines a finite element model
- *History data* defines what happens to the model
  - User divides this history into a sequence of steps, each of which is a period of response of a particular type (static loading, dynamic response, etc.)
  - Definition of a step includes
    - procedure type (static stress,transient heat transfer, etc.)
    - control parameters for time integration or for the nonlinear solution procedures
    - loading
    - output requests

# ABAQUS/Pre

- All data definitions are accomplished with option blocks-sets of data describing a part of the problem definition. Each option is introduced by a keyword line.
- Keyword lines begin with a * in column 1, followed by the option name
- ** is a comment
- Keyword lines are comma separated (free format)
- Data lines may be associated with the option; if so they must follow the keyword line

# ABAQUS/Pre - Model Data

- *NODE and *ELEMENT define individual entities
- *NGEN and *ELGEN generate a range of entities
- Nodes and elements can be grouped into sets
  - NSET and ELSET keyword parameter (comma separated lists)
  - Groups of elements or nodes can then be referred to as one entity
- Material definition (*MATERIAL)
  - All keywords that follow the *MATERIAL option and introduce material behavior relate to the given material until a keyword appears that does not define material properties.
- Boundary conditions (*BOUNDARY)
  - Degrees of freedom
    - 1,2,3 - displacement components
    - 4,5,6 - rotation components
    - Other variables are available based on element type

# ABAQUS/Pre - History Data

- History data is divided into steps
  - *STEP option begins the step
  - *END STEP closes the step
  - Keyword optional parameter PERTURBATION
- *STATIC vs. *DYNAMIC
- Different types of loading
  - *CLOAD concentrated load
  - *DLOAD distributed load
  - AMPLITUDE keyword option for time varying loads

# ABAQUS/Pre - History Data

Results (.res) file

- Used by ABAQUS/Post for graphical post processing such as deformed shape plots or contour plots
- Used to “restart” or continue an analysis
- Command to request that a restart file be written
  *RESTART, WRITE, FREQUENCY=N, OVERLAY

- ABAQUS will write results to the restart file after each increment at which the increment number is exactly divisible by N and at the end of each step of the analysis
- OVERLAY parameter is used to specify that only one increment should be retained on the restart file per step

# ABAQUS/Pre - History Data

Object database (.odb) file

- Binary file used to store model information and analysis results for use by ABAQUS/Viewer
- Variables to be written to the output database are defined by using the following output variable identifiers

```
*OUTPUT, [FIELD|HISTORY], FREQUENCY
*ELEMENT OUTPUT, VARIABLE=ALL|PRESELECT
*NODE OUTPUT
*CONTACT OUTPUT
*ENERGY OUTPUT
```

- FIELD output is used to generate contour plots, displaced shape plots and symbol plots (large data size, low sample frequency)
- HISTORY output is used to generate X-Y plots (small data size, high sample frequency)

# ABAQUS/Pre - History Data

Data (.dat) file

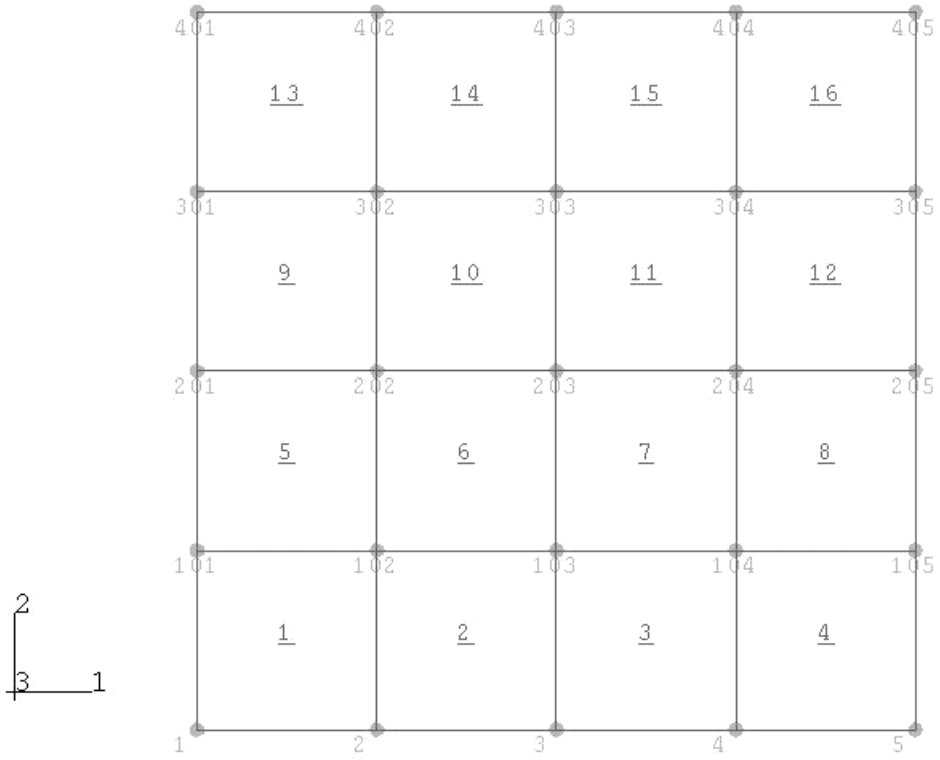
- Text file which contains information about the model definition and tabular output of results.
- The values of output variables can be printed in tabular format throughout the analysis.

```
*EL PRINT
*NODE PRINT
*CONTACT PRINT
*ENERGY PRINT
```

- Specify the variables to be printed in each output table
- For element variables, also specify the locations at which they are to be printed

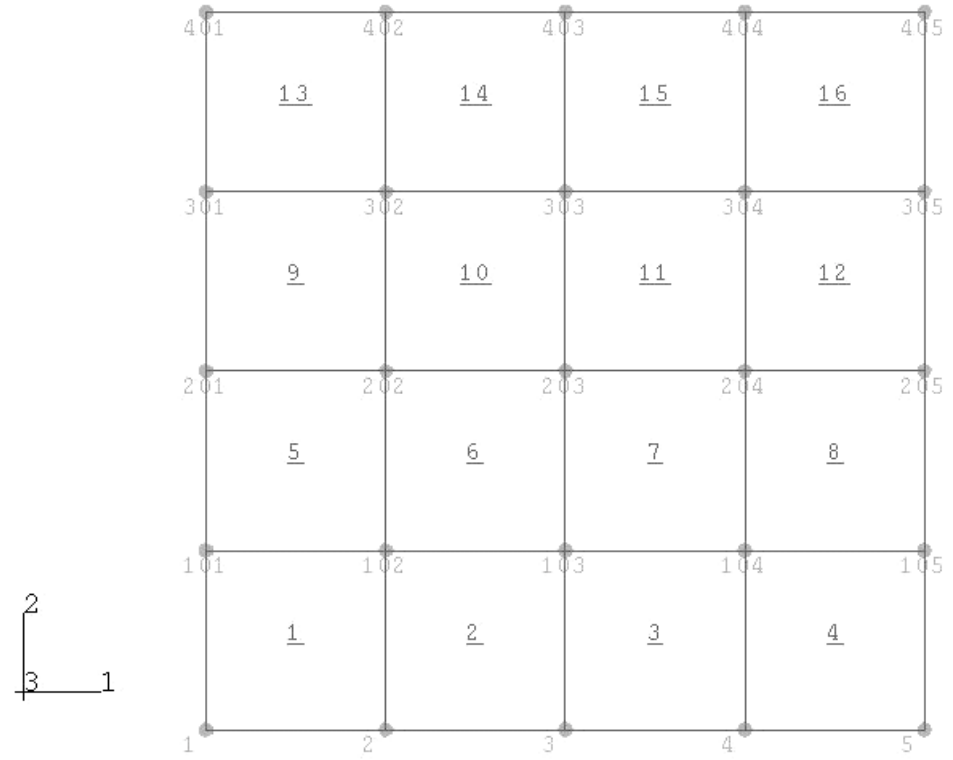
# Sample ABAQUS Input File

```
*HEADING
SIMPLY SUPPORTED SQUARE PLATE WITH UNIFORM PRESSURE---S4R 4 X 4 MESH
*PREPRINT,ECHO=YES,MODEL=NO,HISTORY=NO
*RESTART, WRITE
*NODE, NSET=CORNERS
1,   0.,    0.
5,   1.,    0.
401,  0.,    1.
405,  1.,    1.
*NGEN,NSET=BOT
1,5,1
*NGEN,NSET=TOP
401,405,1
*NFILL
BOT,TOP,4,100
*NSET,NSET=SIDES,GENERATE
1,5,1
401,405,1
1,401,100
5,405,100
```



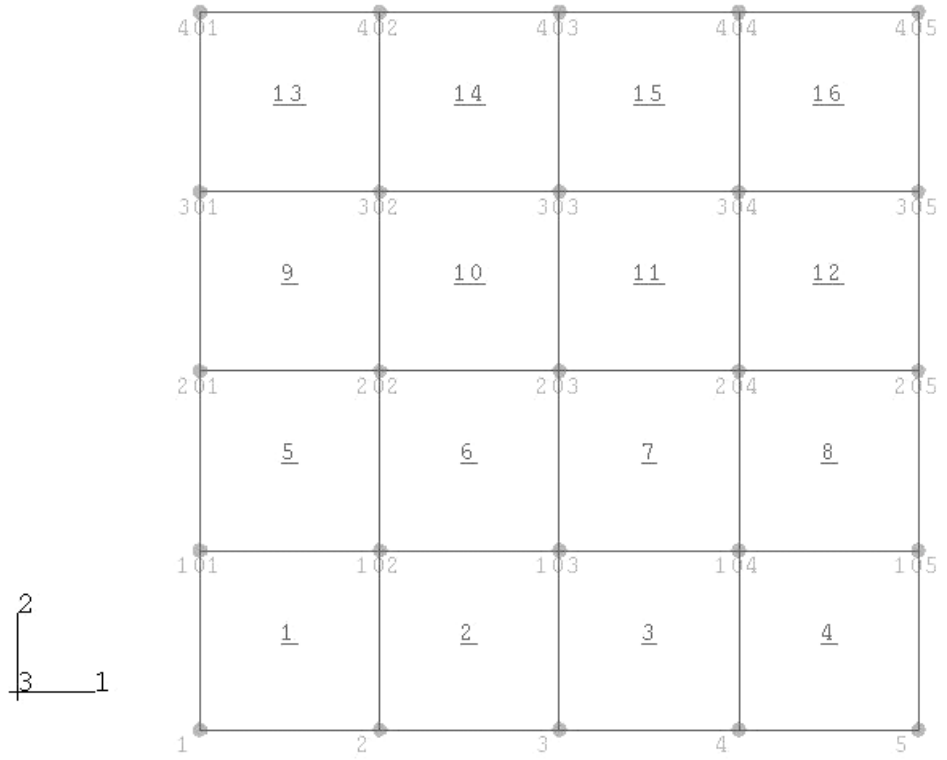
# Sample ABAQUS Input File

```
*NSET,NSET=CENTND
203
*ELEMENT, TYPE=S4R
1, 1, 2,  102, 101
*ELGEN,ELSET=ALLELS
1,4,1,1,4,100,4
*ELSET,ELSET=CENTER
6,7,10,11
*MATERIAL ,NAME=METAL
*ELASTIC
3.E7,  .3
*SHELL SECTION,MATERIAL=METAL,ELSET=ALLEL
.01,3
*BOUNDARY
SIDES,1,3
```



# Sample ABAQUS Input File

```
*STEP
*STATIC
*DLOAD
ALLELS,P,-1.
*NODE PRINT, NSET=CENTND
U
*EL PRINT, ELSET=CENTER, POSITION=AVERAGED
SF
*NODE FILE, NSET=CENTND
U
*EL FILE, ELSET=CENTER, POSITION=AVERAGED
SF
*END STEP
```



# Running ABAQUS Solver on Origin 2000

```
abaqus -v[5.7|5.8-1|5.8-17] interactive [viewer|post|datacheck] job=jobid
```

where jobid =name of input file without extension

Required:

- `interactive` keyword
- Job name

Strongly suggested:

- Code version desired

## Batch submission file

```
#QSUB -r s4r_4x4
#QSUB -lM 16Mw
#QSUB -lT 1:00:00
#QSUB -eo
set -x
cp ~jimg/examples/abaqus/s4r_4x4/s4r_4x4.inp $TMPDIR
cd $TMPDIR
ssusage abaqus -v5.8-17 interactive job=s4r_4x4
cp * ~jimg/examples/abaqus/s4r_4x4
```

# Running datacheck Prior to Solver

```
[origin]$ abaqus -v5.8-1 datacheck interactive job=s4r_4x4
ABAQUS JOB s4r_4x4
BEGIN USER INPUT PROCESSING
Wed May 19 00:55:16 EDT 1999
Run /local/abaqus/5.8/bin/pre.x
ABAQUS License Server checked out 5 Network Tokens
Wed May 19 00:55:19 EDT 1999
END OF USER INPUT PROCESSING
BEGIN IMPLICIT DATACHECK
Wed May 19 00:55:19 EDT 1999
Run /local/abaqus/5.8/bin/standard.x
ABAQUS License Server checked out 5 Network Tokens
Wed May 19 00:55:21 EDT 1999
END OF DATACHECK
ABAQUS JOB s4r_4x4 COMPLETED
```

## Resulting Data Files

```
-rw-r--r--    1    57456 May 19 00:55 s4r_4x4.023
-rwxr-----    1     2392 May 19 00:55 s4r_4x4.com
-rw-r--r--    1    11789 May 19 00:55 s4r_4x4.dat
-rw-r--r--    1     4104 May 19 00:55 s4r_4x4.fil
-rw-r-----    1      795 May 19 00:53 s4r_4x4.inp
-rw-r--r--    1     1186 May 19 00:55 s4r_4x4.msg
-rw-r--r--    1    62968 May 19 00:55 s4r_4x4.odb
-rw-r--r--    1    98304 May 19 00:55 s4r_4x4.res
-rw-r-----    1    49524 May 19 00:55 s4r_4x4.sdb
```

# Running ABAQUS Solver on T90

```
[OSCA]$ abaqus -v5.7 job=s4r_4x4
ACADEMIC ABAQUS 5.7-1 Network --- CRAY
JOB TIME LIMIT ?
 enter s for s secs.
 enter m:s for m mins., s secs.
 enter h:m:s for hrs, mins., secs.
 or carriage return for 1:0    : 120
 enter m for m megawords or
 carriage return for 8 Mw : 8
ABAQUS/STANDARD is running on a Category D machine.
nqs-181 qsub: INFO
  Request <7567.osca>: Submitted to queue <batch> by <jimg(6815)>.
```

## Resulting Data Files

```
-rw-r--r--   1        0 May 19 02:07 abaqus.env
-rwxr-----   1     2393 May 19 02:07 s4r_4x4.com
-rw-r--r--   1     2363 May 19 02:07 s4r_4x4.dat
-rw-r-----   1      795 May 19 01:50 s4r_4x4.inp
-rw-r--r--   1     4960 May 19 02:07 s4r_4x4.log
-rw-r--r--   1     2854 May 19 02:07 s4r_4x4.msg
-rw-r--r--   1       70 May 19 02:07 s4r_4x4.q
```

# Viewing Results with ABAQUS/Post

- ABAQUS/Post is being replaced by ABAQUS/Viewer
- Text command interface
- Mouse input for image rotation, displacement and zoom

```
abaqus -v5.8 post restart=s4r_4x4
```

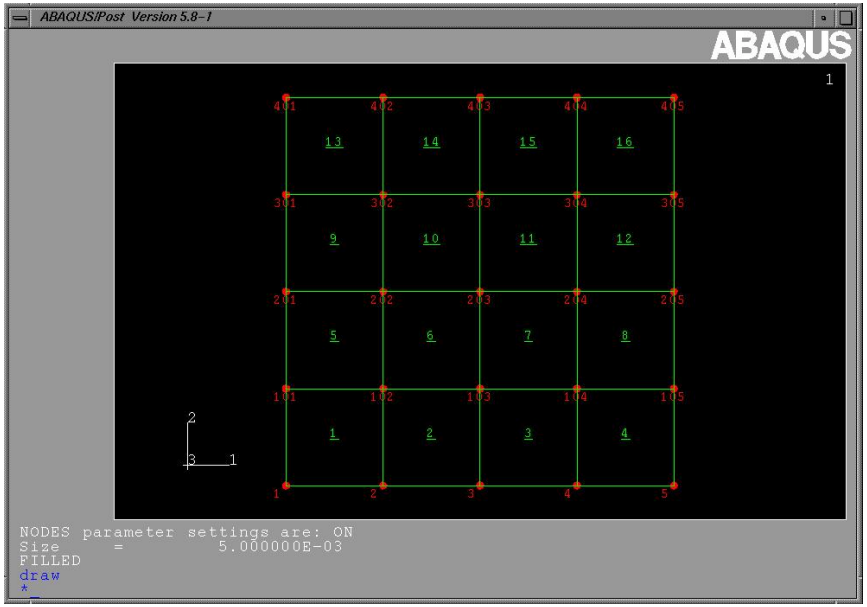
```
Reading restart file : s4r_4x4

      ABAQUS Version: 5.8-1          Date: 12-MAY-1999

      No. of elements     :      16
      No. of nodes        :      25
      No. of element sets :       3
      No. of node sets    :       6

  Reading element data...
  Reading node data...
  Reading element sets...
  Reading node sets...

  Restart file = s4r_4x4
  Current step/inc = 1 / 1
```



# ABAQUS/Post Commands

`draw, options`

- `displaced` - original grid and deformed grid overlaid in different colors

`contour, options`

- `variable=parameter` (required) - name of the variable to be contoured
- `range` - print the min and max values of this variable, without plotting

`set, options`

- `underformed`=[on|off] - command to plot only the deformed shape
- `d magnification=#` - scale displacements to improve display
- `n numbers=[on|off]` - option to display node numbers
- `el numbers=[on|off]` - option to display element numbers
- `nodes=#` - control the size of dot symbol displayed at node locations
- `bc display=[on|off]` - toggles the plotting of boundary conditions
- `load display=[on|off]` - toggles the plotting of loads
- `fill=[on|off]` - contour plots with discrete filled colored regions
- `shade=[on|off]` - contour plots with smooth shaded colored regions

# ABAQUS/Post Commands

`report elements, parameter`

Report the element number, element type, material name (if applicable) and nodal connectivity list

`displaced` - this parameter indicates the displaced shape is to be used to select the elements for reporting

`report nodes, parameter`

Report the node number, the current coordinates, and, if available, the displacements for nodes picked

`displaced` - see description above

`original` - use this parameter if the reported coordinates will be those of the undeformed shape

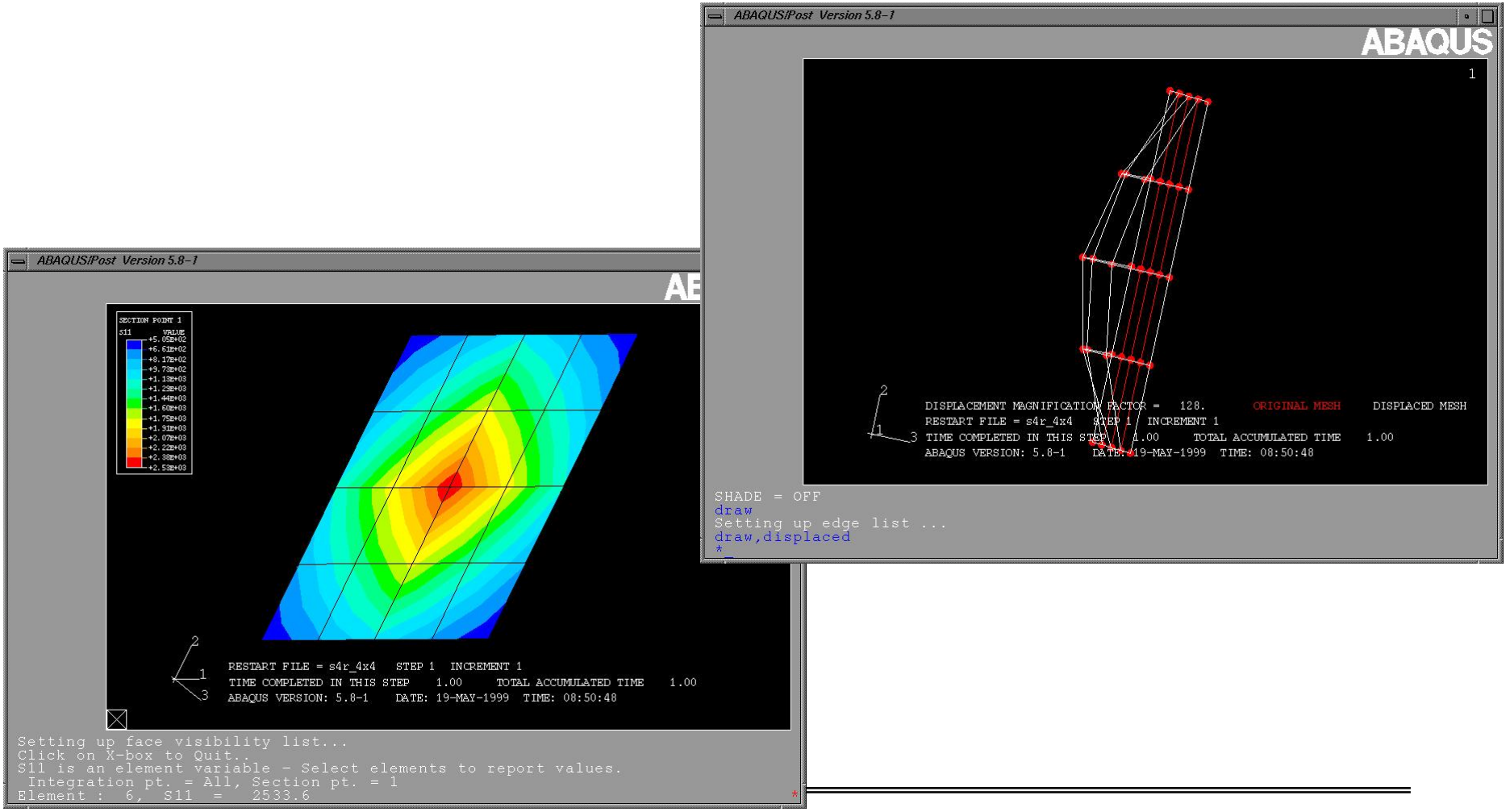
`report values, parameter`

`variable=name`(required) - name of the variable to be reported

`displaced` - see description above

`integration point` - set this parameter equal to the element integration point at which the variable is to be obtained; default is all integration points

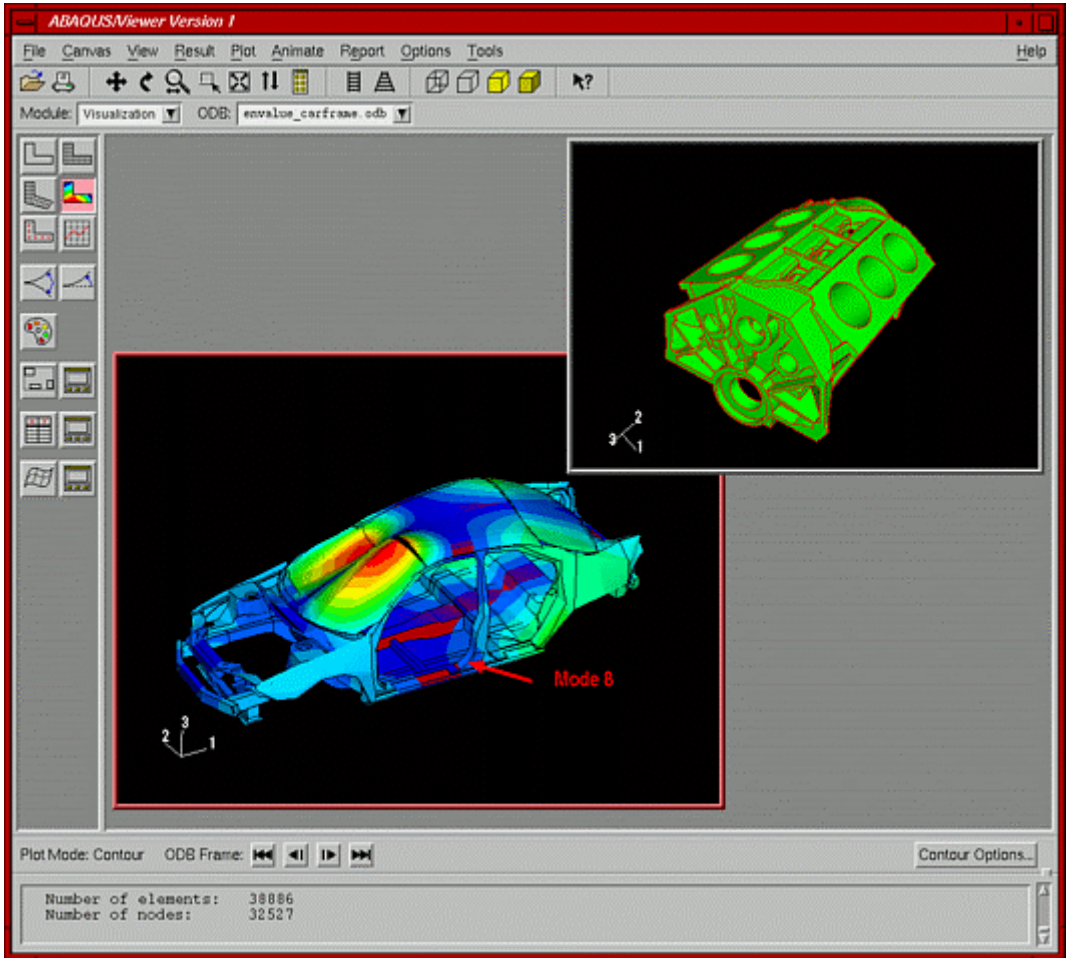
# ABAQUS/Post Example Problem



# Viewing Results with ABAQUS/Viewer

- All of the functionality of ABAQUS/Post with nice GUI interface
- Will replace ABAQUS/Post after v5.8
- What “replace” actually means is yet to be determined

```
abaqus -v5810 viewer odb=jobname
```


ABAQUS/Viewer Version 1
File Canvas View Result Plot Animate Report Options Tools Help
Module: Visualization ODB: envalue_carframe.odb
Mode 8
Plot Mode: Contour ODB Frame: Contour Options...
Number of elements: 38886
Number of nodes: 32527

# ABAQUS Web Information

- ABAQUS Home page:
  http://www.abaqus.com
- ABAQUS Examples:
  http://www.abaqus.com/applications/applications.html
- ABAQUS Training:
  http://www.abaqus.com/training-services/training-services.html
- Official OSC supported software page:
  http://oscinfo.osc.edu/software

# ANSYS Package

ANSYS Modeling Features

ANSYS Components

ANSYS Documentation

Model Development

Running ANSYS Solver on T90

Running ANSYS Solver on Origin

ANSYS Post Processing

# ANSYS Modeling Features

Nonlinear and linear analysis

Structural
- static, dynamic, buckling, kinematic

Thermal
- steady-state, transient, phase change, radiation

Acoustics

Coupled field
- Thermal-structural
- Acoustics-structural
- Thermal-electric
- Piezoelectrics

Materials
- Plasticity
- Hyperelasticity
- Viscoplasticity
- Viscoelasticity
- Creep, swelling
- Temperature-dependent properties
- Phase change via enthalpy

Elements
- Quadrilateral, triangle, mixed quad/triangle, mapped brick and tetrahedral
- Cables and compression only spars
- Surface-to-surface contact w/friction
- Node-to-node contact with friction
- Cracking and/or crushing solids
- Element birth and death
- Substructuring for linear regions
- Thermal-electric shell

# ANSYS Components

- Solver, Pre and Post processor are integrated into one package
- Preprocessor
  - Motif-based windows look and feel
  - Cascading pull-down menus
  - Online-documentation and hyper-text-linked help
  - Automatic FE mesh generation
  - Adaptive, mapped meshing
  - P- and h- element meshing
- Postprocessing
  - Contours, vector displays and graphics of potentials and field data
  - Animation of results data
  - 3D volume visualization tools including isosurfaces, gradient displays, volume slicing, section displays and translucency
  - General line integral and mapping features

# ANSYS Documentation

- Hardcopy manuals can be purchased from HKS, Inc.
- Full on-line documentation is available:

```
module load ansys
anshelp55
```

ANSYS Basic Analysis Procedures Guide
ANSYS Modeling and Meshing Guide
ANSYS Structural Analysis Guide
ANSYS Thermal Analysis Guide
ANSYS Coupled-Field Analysis Guide
ANSYS Advanced Analysis Techniques
ANSYS Commands Manual
ANSYS Elements Manual
ANSYS Theory Manual
ANSYS Operations Guide

- Hypertext-based help system is also available via “HELP” buttons during the analysis

# General Analysis Procedure

Assemble model - graphical interface

- Select element types and set options
- Define real constants
- Define material properties
- Generate model geometry
- Mesh geometry
- Apply boundary conditions
- Apply loads

Run solver - batch mode

View results - graphical interface

- Deformed shape plot
- Contour plots
- X-Y data plots

# ANSYS Command Line Options

```
ansys -m MW -j jobname -g
```

- `-m MW`: This option specifies the amount of working storage obtained from the system. The units are megawords. The memory requirement for the entire execution will be approximately 5.3 megawords more than the -m specification.
- `-j jobname`: This option allows the user to specify the jobname on the command line. When loading data, ANSYS uses the jobname as the file name and looks for specific extensions.
- `-g`: Specifies that the ANSYS menu system be started automatically.

ANSYS requires the `module load` command to set proper environment variables

```
module load ansys
```

# ANSYS User Interface

## Main Menu

•Primary ANSYS functions

•Pop-up menus based on the progression of the program

## Utility Menu

•Pull-down menus

•Available any time during analysis

## Input Window

•Provides area for typing in ANSYS commands

•Command history

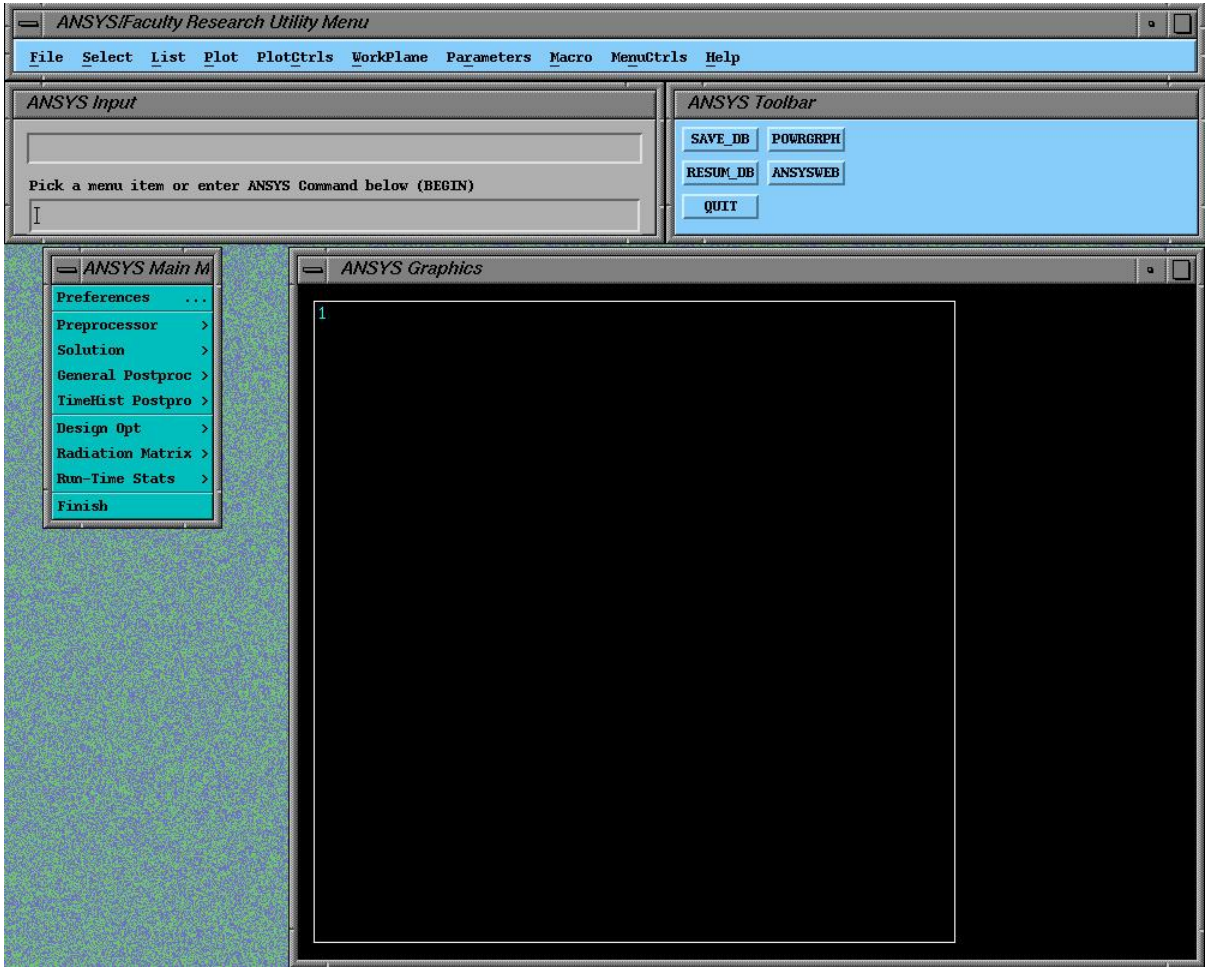
## Graphics Window

•Area for graphics display

## Toolbar

•One-click access to common functions

•User can customize the toolbar

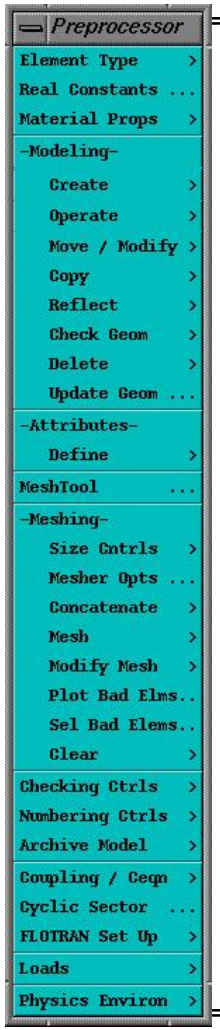


# Model Development

**Preprocessor**

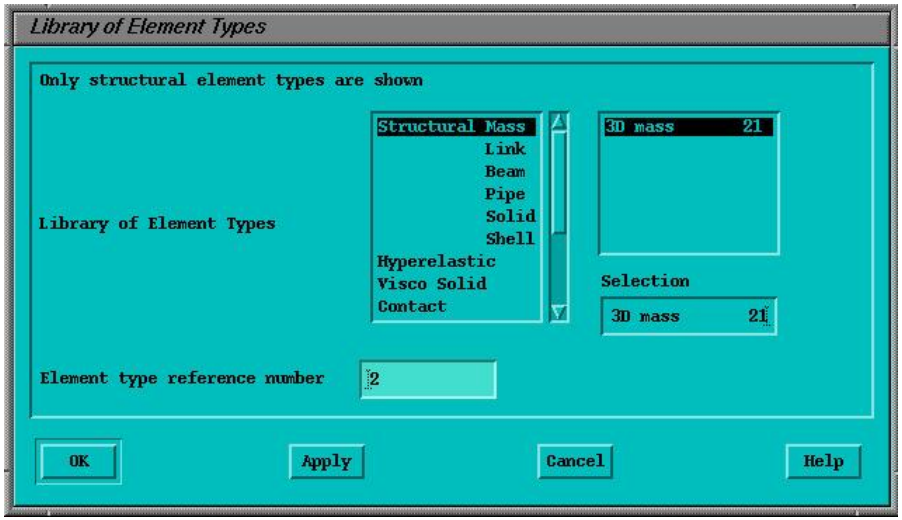
- ANSYS Main Menu will be the starting point for each phase of the analysis
- **Preferences** allow you to specify what type of analysis you are performing
- This will mask out unwanted menu items
- "Preprocessor" selection brings up the menu on the right

# Selecting Elements in ANSYS



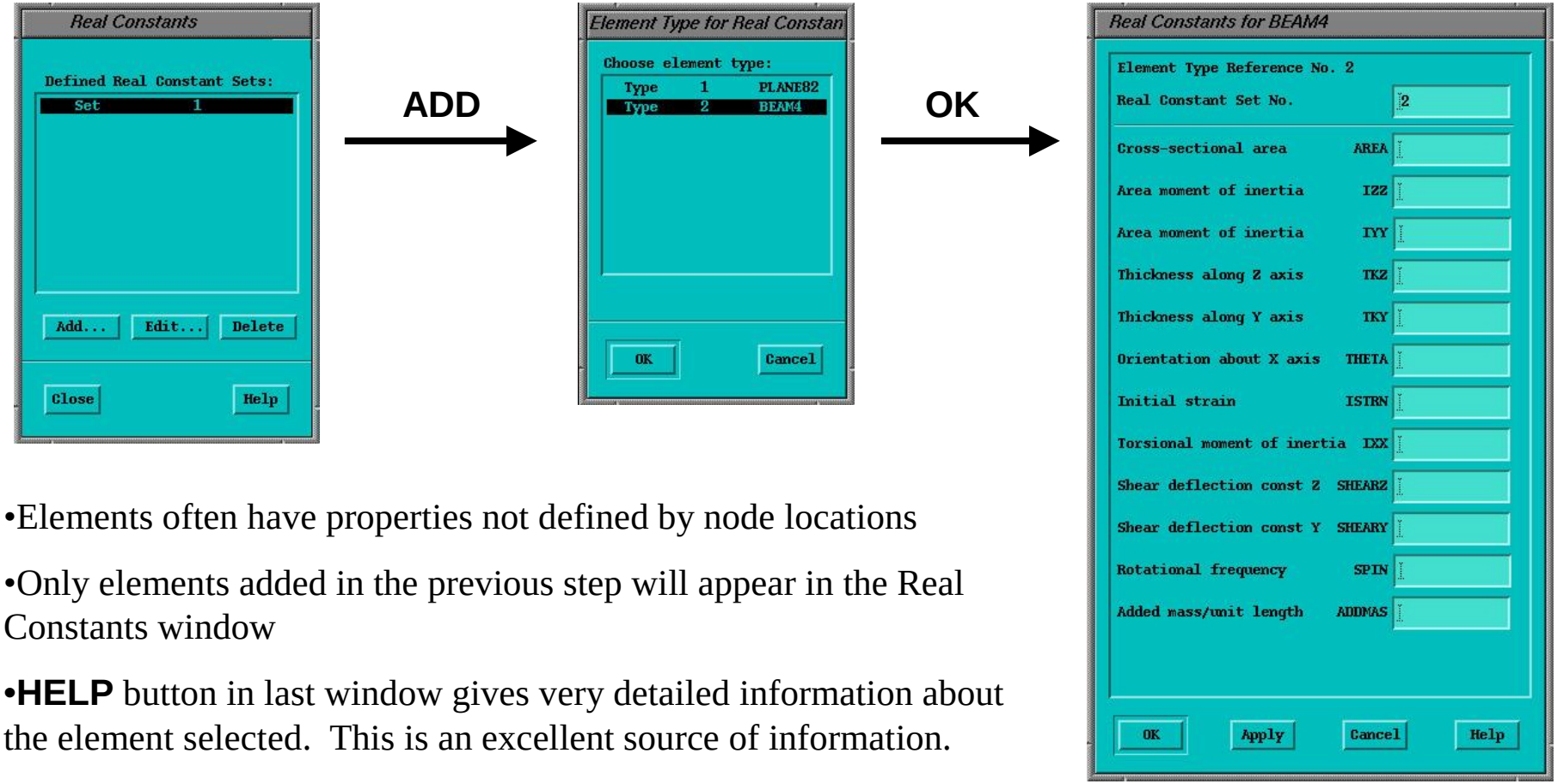

- Elements are first added to the "active" list
- Material and other physical properties are assigned
- During the meshing procedure, "active" elements can be selected

Choosing **ELEMENT TYPE** from the preprocessor window allows you select the element(s) you want to add



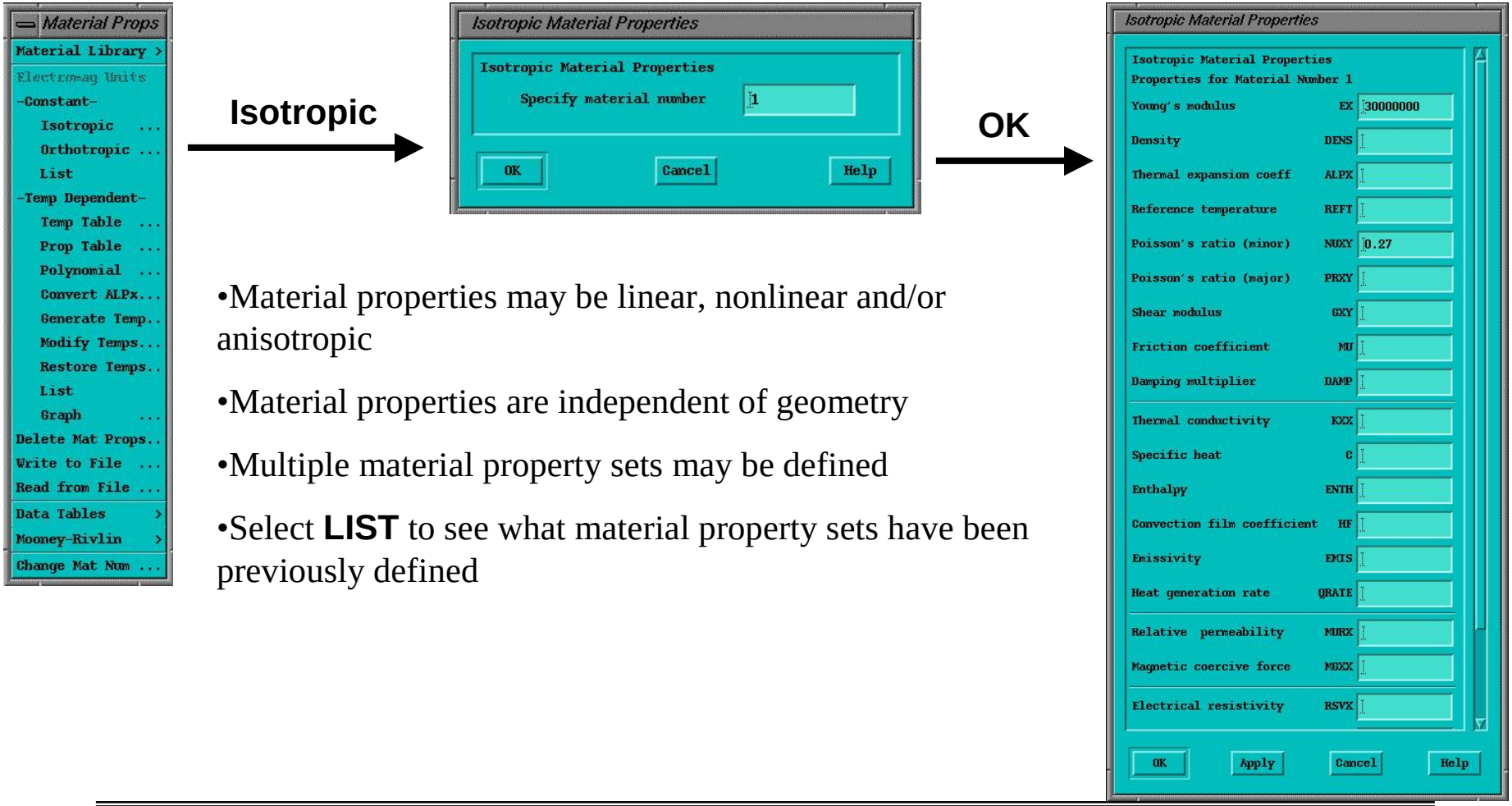

Once an element is added, you must also define **OPTIONS** for the particular element type

# Defining Real Constants



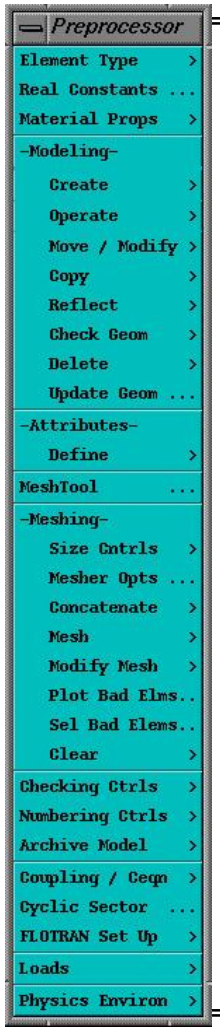

- Elements often have properties not defined by node locations
- Only elements added in the previous step will appear in the Real Constants window
- **HELP** button in last window gives very detailed information about the element selected. This is an excellent source of information.

# Material Properties



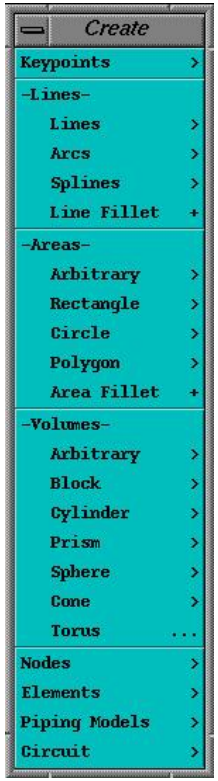

- Material properties may be linear, nonlinear and/or anisotropic
- Material properties are independent of geometry
- Multiple material property sets may be defined
- Select **LIST** to see what material property sets have been previously defined

# Creating Model Geometry

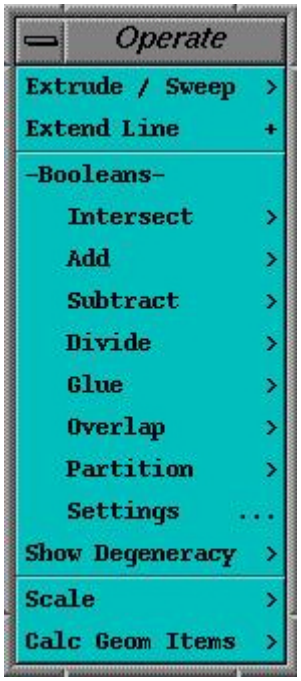



- ANSYS interface has many CAD features
- General procedure
  - Build up basic model with circles, rectangles
  - Create more complex areas with lines and splines
  - Refine model with rounding and fillets
  - Subtract, combine and/or add areas
- You can create points, lines, areas and volumes

**Create**

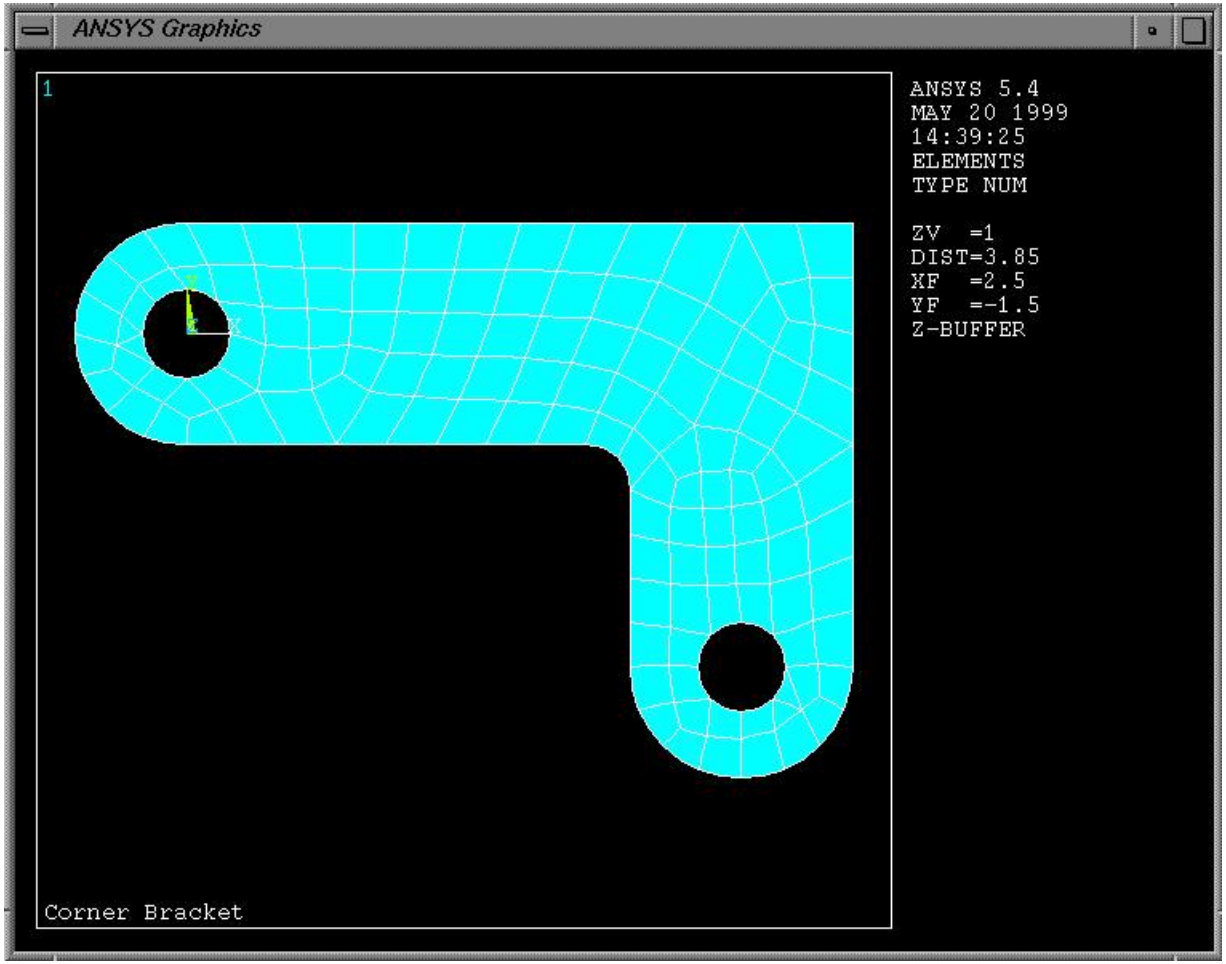


# Modeling Operate Functions

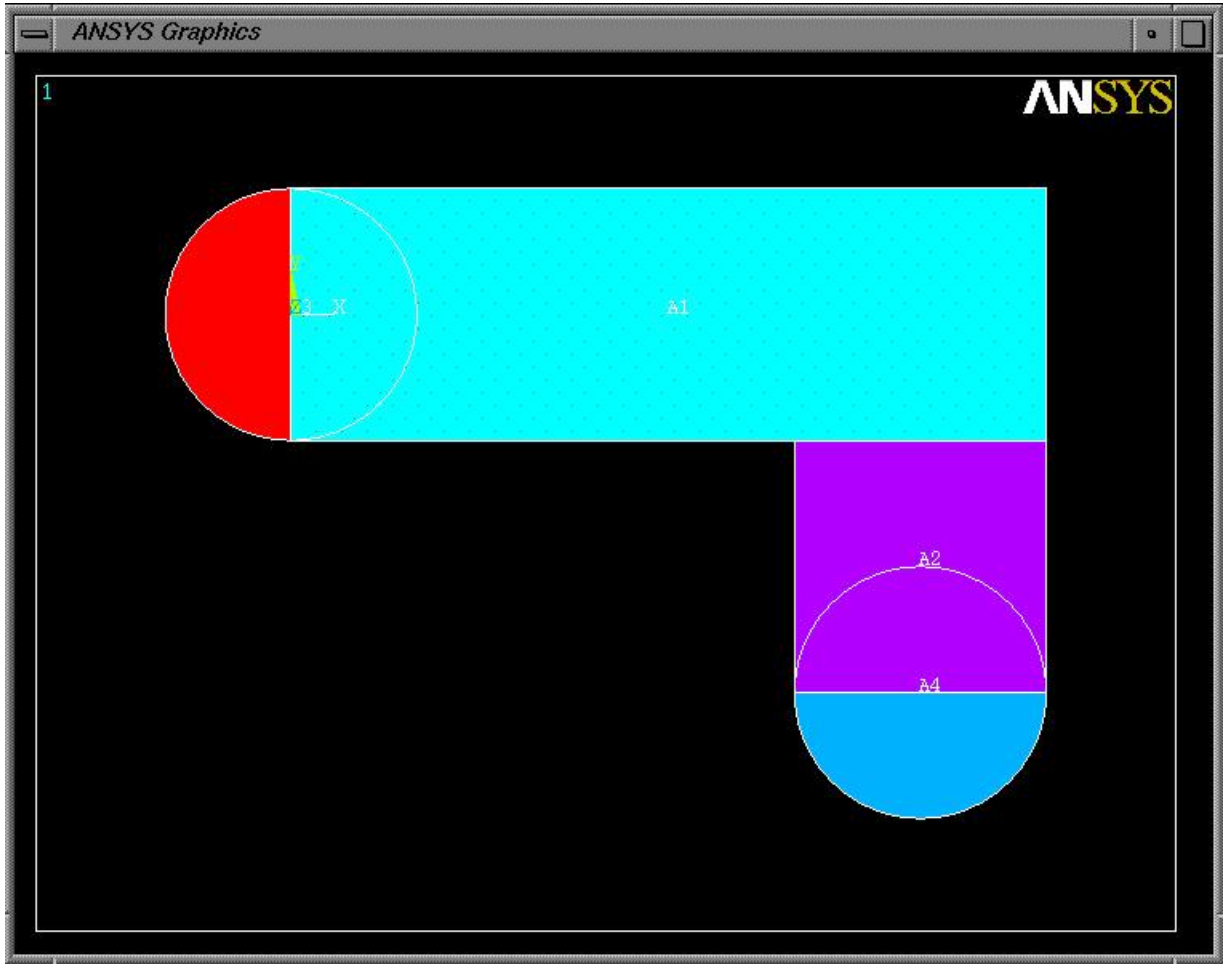



- Powerful features that allow you to create complex geometry from simpler objects
- Intersect generates an area from the overlapping parts of two areas
- Add combines multiple areas into one
- Subtract allows you to create voids in a solid area

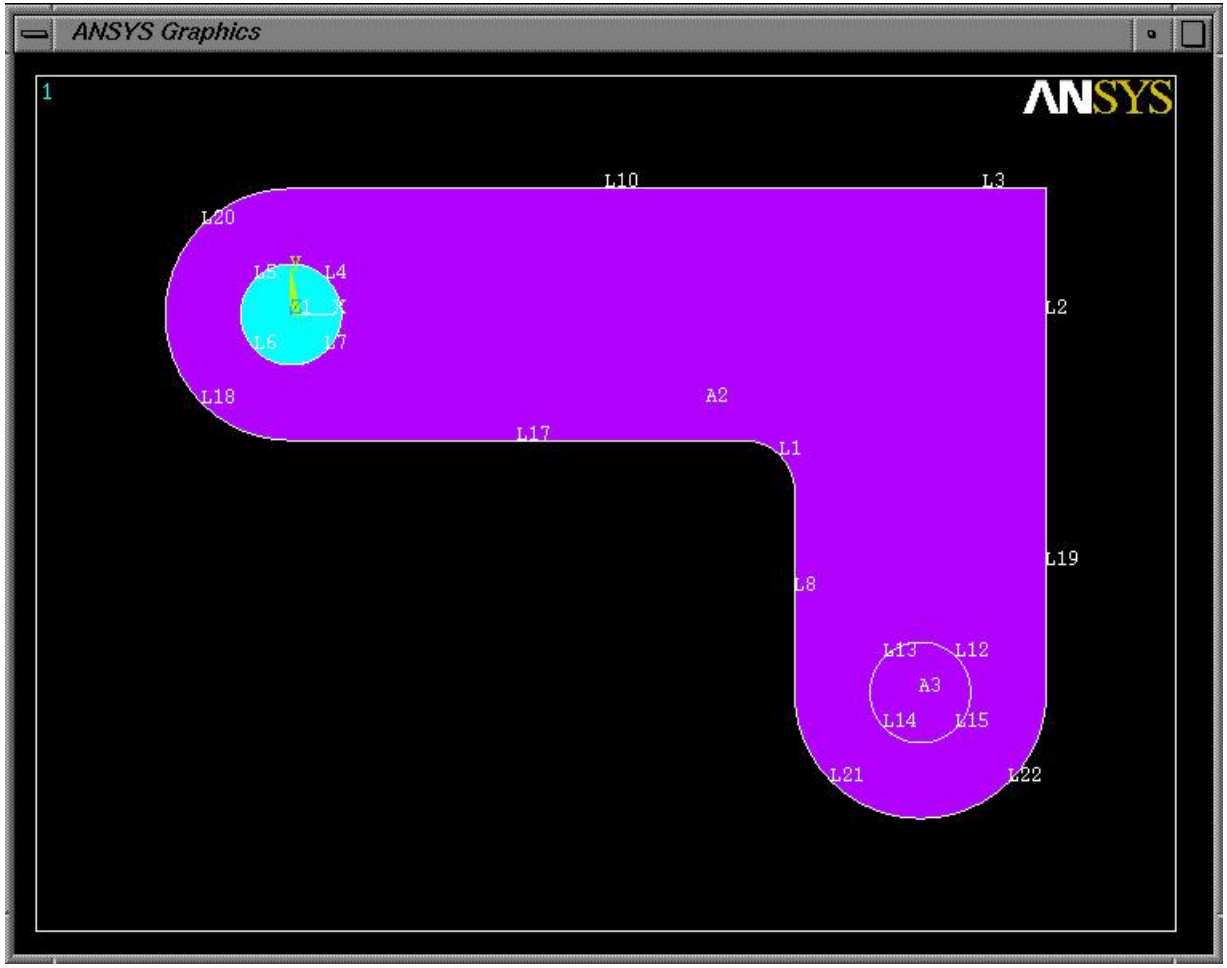
# Corner Bracket Example



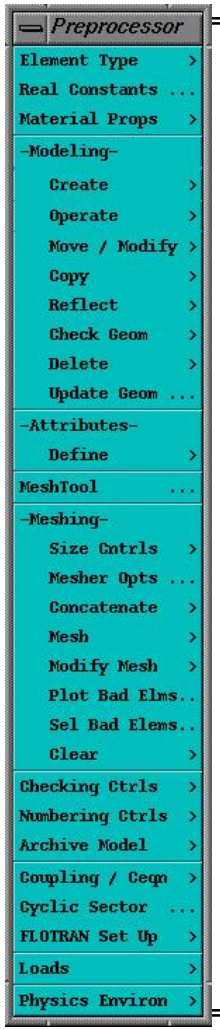
# Corner Bracket Example



# Corner Bracket Example

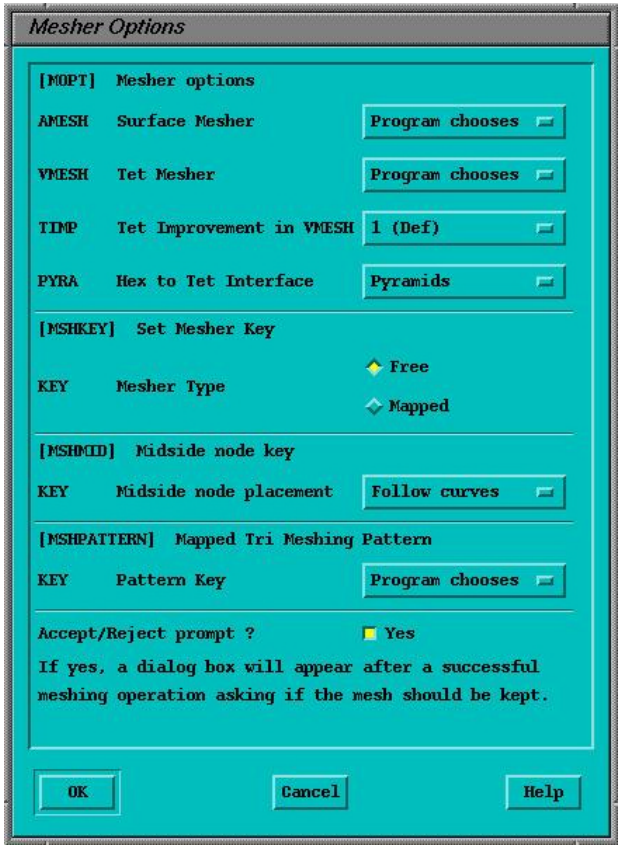


# Meshing

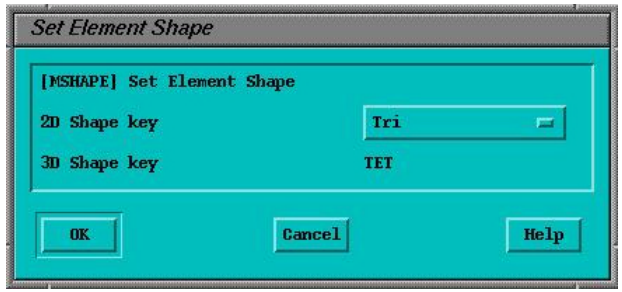



**Mesher Options**

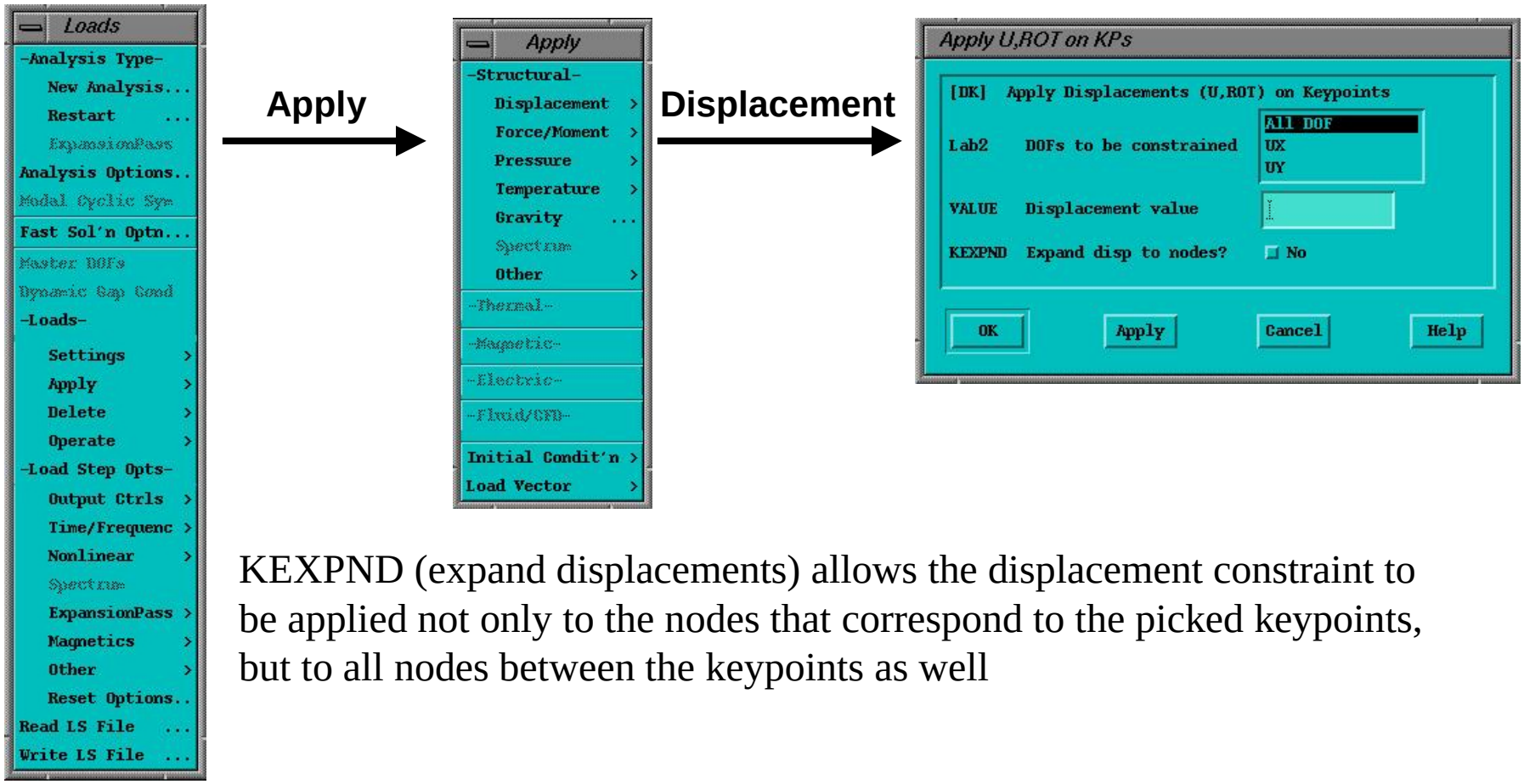
- Use automatic meshing for first cut
- Refine using **MeshTool** if needed
- Suggest turning on Accept/Reject prompt
- Use **Size Controls** option to control size of elements in initial mesh




OK



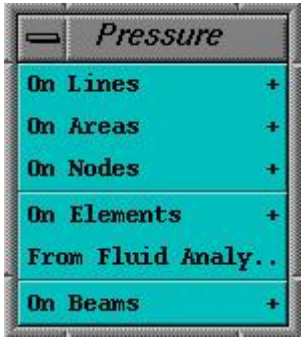
# Loads and Boundary Conditions




KEXPND (expand displacements) allows the displacement constraint to be applied not only to the nodes that correspond to the picked keypoints, but to all nodes between the keypoints as well

# Loads and Boundary Conditions

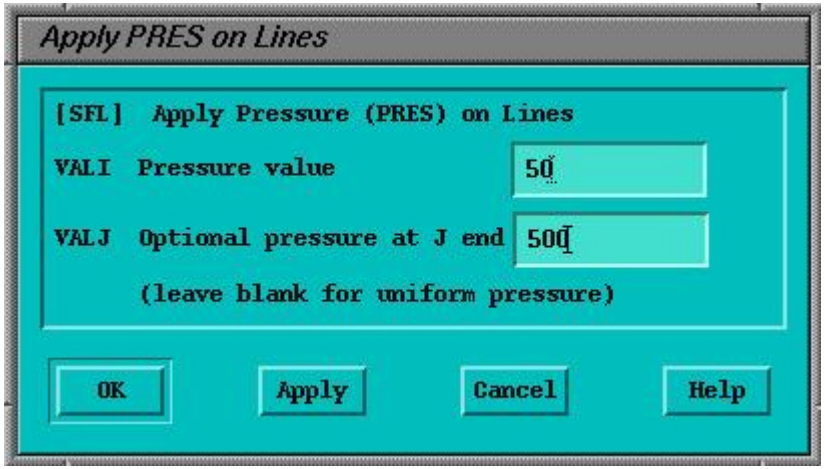
**Pressure** →




**On Lines** →

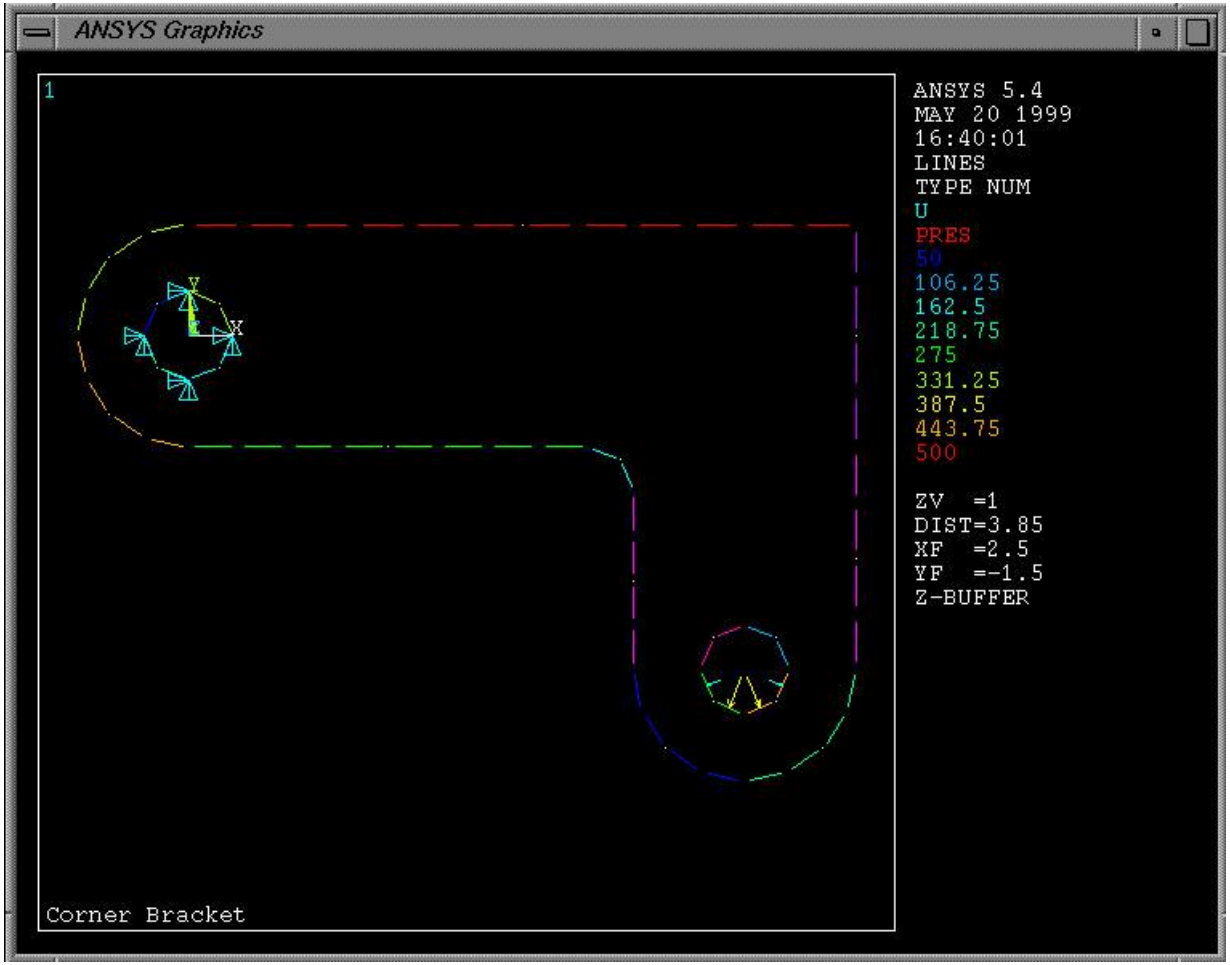
By various options, pick the line you want to apply the pressure load to

↓




- Entering different values for VALI and VALJ will apply a linear pressure load between the endpoints of the line
- Pressure is always positive into the material

# Loads and Boundary Conditions



# Running ANSYS Solver on T90

- Environment variables set up with the `module load` command
- `ja` command provides a detailed resource usage report including maximum memory used
- Must specify a minimum of 17Mw
- `commands.txt` file contains the actual commands that are sent to the ANSYS program

Batch submission file

```
#QSUB -r ansys
#QSUB -lT 300
#QSUB -lM 20Mw
#QSUB -eo
set -x
module load ansys
cp $HOME/examples/ansys/* $TMPDIR
ja
ansys -j bracket < commands.txt
ja -cst
cp * $HOME/examples/ansys
```

commands.txt

```
RESUME
/SOLU
CHECK
SOLVE
FINISH
/EXIT,ALL
```

# Running ANSYS Solver on Origin

- Environment variables set up with the `module load` command
- `ssusage` command provides resource usage report including maximum memory used
- `commands.txt` file contains the actual commands that are sent to the ANSYS program

Batch submission file

```
#QSUB -r ansys
#QSUB -lT 300
#QSUB -lM 20Mw
#QSUB -eo
set -x
module load ansys
cp $HOME/examples/ansys/* $TMPDIR
ssusage ansys -j bracket < commands.txt
cp * $HOME/examples/ansys
```

commands.txt

```
RESUME
/SOLU
CHECK
SOLVE
FINISH
/EXIT,ALL
```

# ANSYS Postprocessing

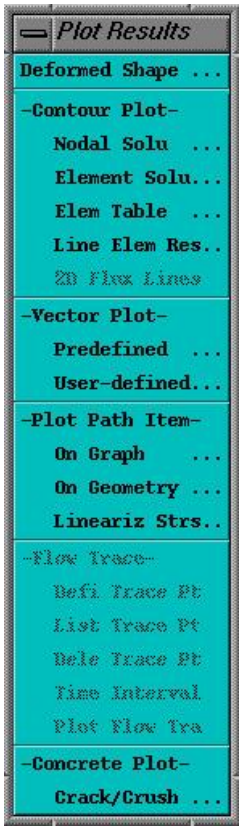
- After batch job completes, review log file for errors
- Restart ANSYS in graphical interface mode with the correct job name

```
ansys -g -j bracket
```

- From the TOOLBAR, click RESUME_DB
  or
  From the UTILITY MENU, click RESUME JOBNAME.DB from the FILE pull down menu

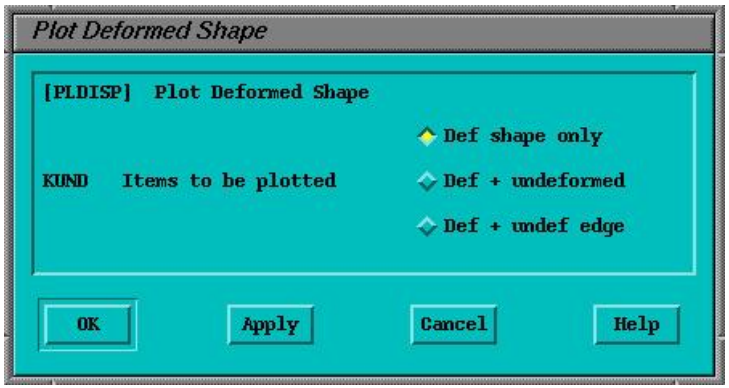
# ANSYS Postprocessing

**Plot Results**



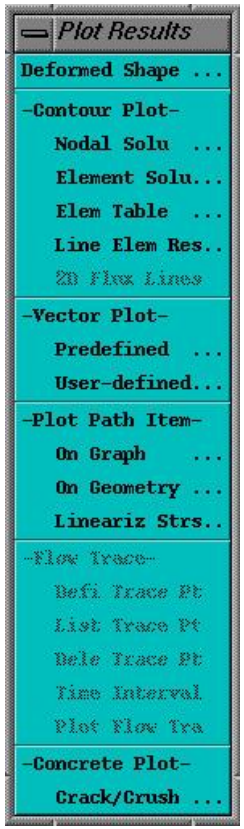

- After solution is obtained, you must read in the results by selecting one of the options under **READ RESULTS**
- Results fall into 4 major categories
  - Deformed shape plots
  - Contour plots
  - X-Y plots
  - Viewing numerical results

**Deformed Shape**

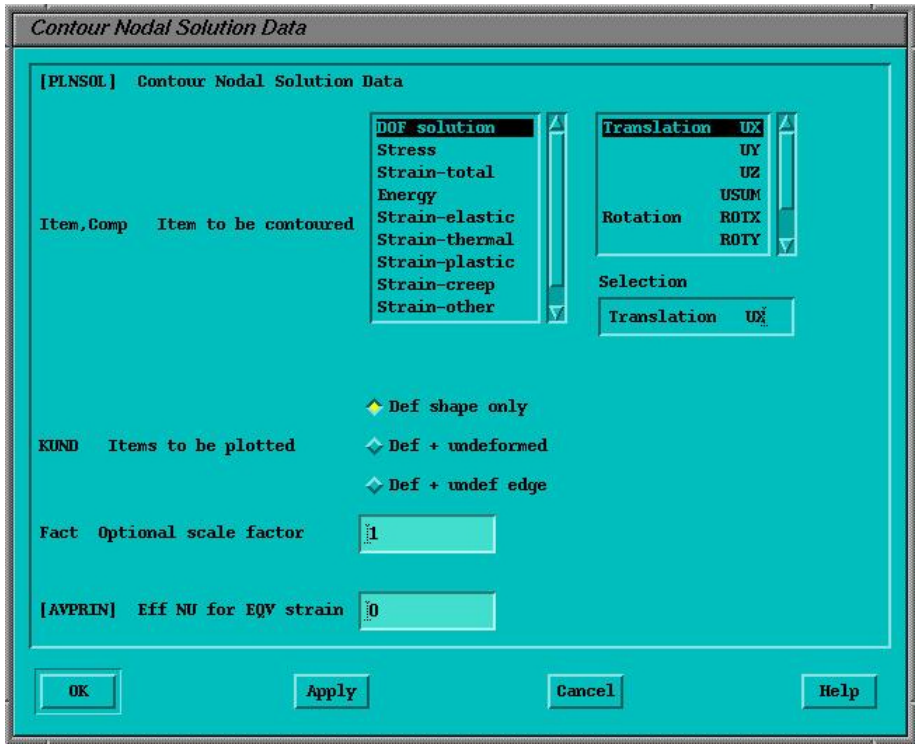


# Contour Plots

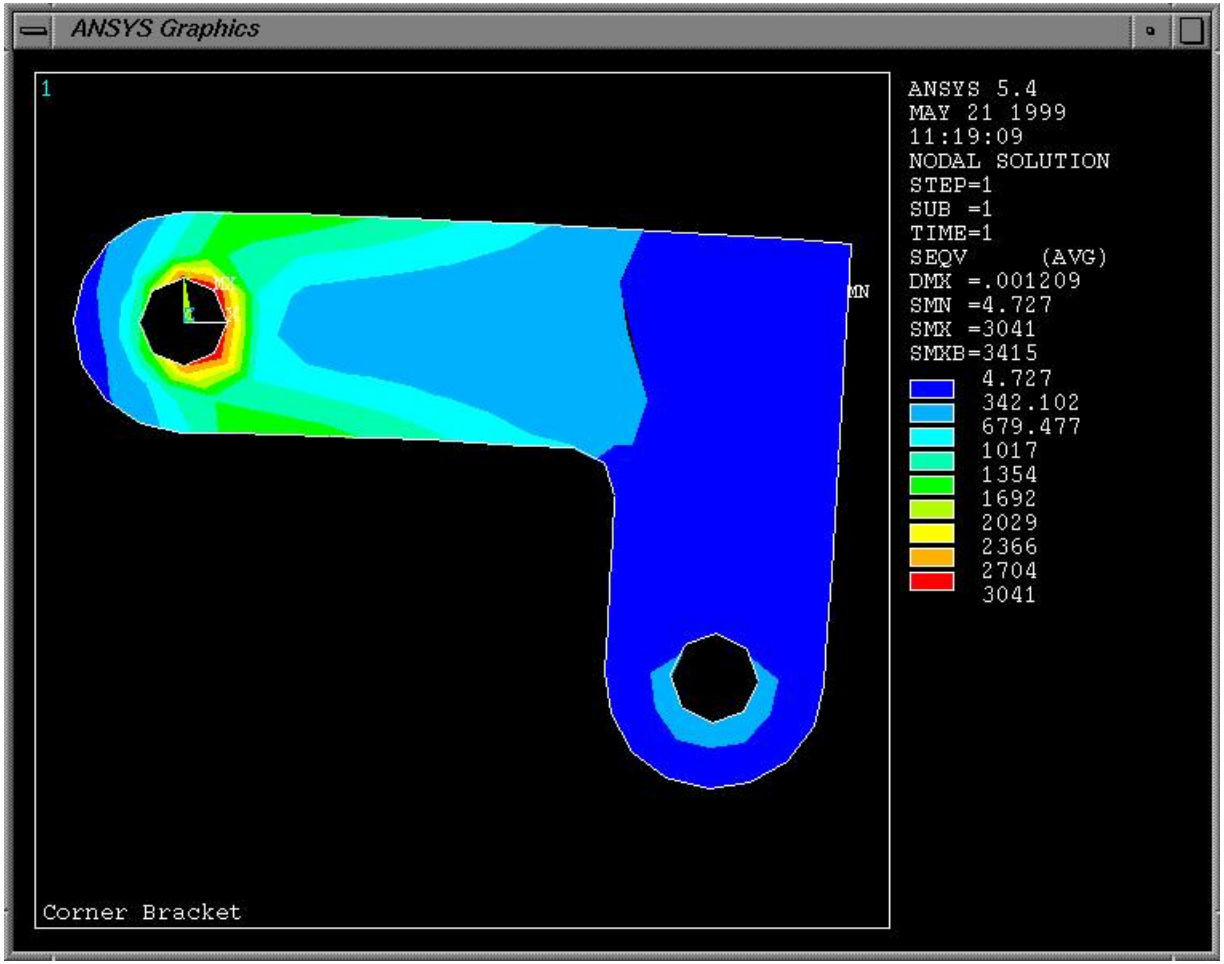
- Contours available for both nodal and element properties
- All parameters saved, no need to specify before the solution which parameters you would like to examine




**Nodal Soln**

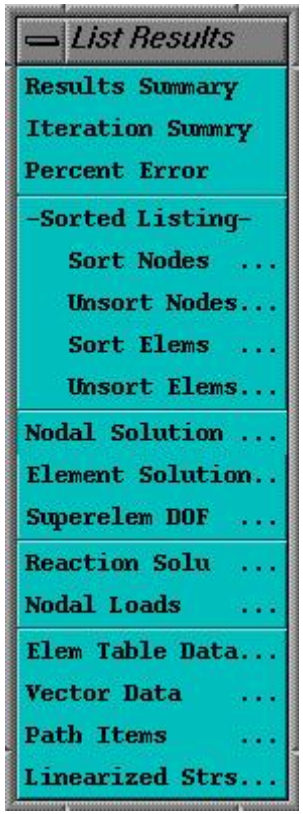


# Contour Plots

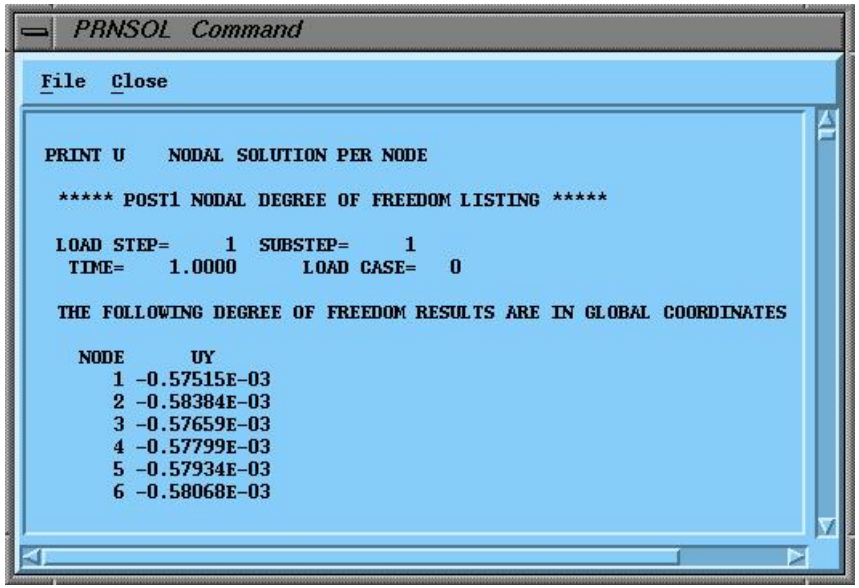


# Viewing Numerical Results

**List Results**




- Procedure similar to contour plots
- Rather than a plot appearing, a text window will pop up with tabulated data within



# ANSYS Web Information

- ANSYS home page:
  http://www.ansys.com
- List of ANSYS documentation:
  http://www.ansys.com/ServSupp/Library/library.html
- Vendor training:
  http://www.ansys.com/ServSupp/Training/index.html
- "ANALYSIS SOLUTIONS", an independent magazine covering design analysis and optimization for ANSYS users:
  http://www.analysismag.com
- Downloadable information:
  http://www.ansys.com/Download
- Official OSC supported software page:
  http://oscinfo.osc.edu/software